IBM Planning Analytics
2.0

# *Planning Analytics Workspace*



**Note**

Before you use this information and the product it supports, read the information in “Notices” on page 1271.

## Product Information

This document applies to IBM Planning Analytics Version 2.0 and might also apply to subsequent releases.

Licensed Materials - Property of IBM

Last updated: 2025-10-09

**© Copyright International Business Machines Corporation 2009, 2025.**
US Government Users Restricted Rights – Use, duplication or disclosure restricted by GSA ADP Schedule Contract with IBM Corp.

# Contents

**Introduction** ..... **xvii**

**Chapter 1. What's new in Planning Analytics Workspace** ..... **1**

2.0.107 - What's new, September 24, 2025 ..... 1
What's new in books and reporting - 2.0.107 ..... 1
What's new in plans and applications - 2.0.107 ..... 2
What's new in modeling - 2.0.107 ..... 3
What's new in administration - 2.0.107 ..... 3
2.0.106 - What's new, July 29, 2025 ..... 4
What's new in books and reporting - 2.0.106 ..... 4
What's new in plans and applications - 2.0.106 ..... 6
What's new in modeling - 2.0.106 ..... 7
What's new in administration - 2.0.106 ..... 9
Known issues - 2.0.106 ..... 9
2.0.105 - What's new, June 25, 2025 ..... 10
What's new in books and reporting - 2.0.105 ..... 10
What's new in modeling - 2.0.105 ..... 11
What's new in administration - 2.0.105 ..... 13
2.0.104 - What's new, May 20, 2025 ..... 14
What's new in books and reporting - 2.0.104 ..... 15
What's new in modeling - 2.0.104 ..... 17
What's new in administration - 2.0.104 ..... 17
Known issues - 2.0.104 ..... 18
2.0.103 - What's new, April 22, 2025 ..... 19
What's new in books and reporting - 2.0.103 ..... 19
What's new in modeling - 2.0.103 ..... 22
What's new in administration - 2.0.103 ..... 26
What's new in Planning Analytics Workspace Local - 2.0.1032.1.10 ..... 27
Known issues - 2.0.103 ..... 27
2.0.102 - What's new, March 19, 2025 ..... 28
What's new in books and reporting - 2.0.102 ..... 28
What's new in plans and applications - 2.0.102 ..... 33
What's new in modeling - 2.0.102 ..... 35
2.0.101 - What's new, February 18, 2025 ..... 38
What's new in books and reporting - 2.0.101 ..... 38
What's new in plans and applications - 2.0.101 ..... 53
What's new in modeling - 2.0.101 ..... 55
What's new in administration - 2.0.101 ..... 61
Known issues - 2.0.101 ..... 62
2.0.100 - What's new, December 11, 2024 ..... 62
Important upgrade requirement for Planning Analytics Workspace Local 2.0.100 ..... 62
What's new in books and reporting - 2.0.100 ..... 62
What's new in plans and applications - 2.0.100 ..... 64
What's new in modeling - 2.0.100 ..... 67
2.0.99 - What's new, October 23, 2024 ..... 71
What's new in books and reporting - 2.0.99 ..... 71
What's new in modeling - 2.0.99 ..... 80
What's new in administration - 2.0.99 ..... 86
Known issues - 2.0.99 ..... 86
2.0.98 - What's new, August 21, 2024 ..... 87

What's new in books and reporting - 2.0.99 ... 87
What's new in modeling - 2.0.98 ... 91
What's new in administration - 2.0.98 ... 94
2.0.97 - What's new, July 16, 2024 ... 97
General updates - 2.0.97 ... 97
What's new in books and reporting - 2.0.97 ... 98
What's new in plans and applications - 2.0.97 ... 101
What's new in modeling - 2.0.97 ... 103
What's new in administration - 2.0.97 ... 103
Deprecation notices - 2.0.97 ... 104
2.0.96 - What's new, May 28, 2024 ... 105
What's new in books and reporting - 2.0.96 ... 105
What's new in modeling - 2.0.96 ... 114
What's new in administration - 2.0.96 ... 118
Deprecation notices - 2.0.96 ... 120
2.0.95 - What's new, April 23, 2024 ... 120
What's new in books and reporting - 2.0.95 ... 120
What's new in plans and applications - 2.0.95 ... 123
What's new in modeling - 2.0.95 ... 125
Deprecation notices - 2.0.95 ... 128
2.0.94 - What's new, March 20, 2024 ... 128
What's new in books and reporting - 2.0.94 ... 129
What's new in plans and applications - 2.0.94 ... 132
What's new in modeling - 2.0.94 ... 133
What's new in administration - 2.0.94 ... 134
Deprecation notices - 2.0.94 ... 136
2.0.93 - What's new, February 20, 2024 ... 136
What's new in books and reporting - 2.0.93 ... 136
What's new in plans and applications - 2.0.93 ... 140
What's new in modeling - 2.0.93 ... 141
What's new in administration - 2.0.93 ... 144
Deprecation notices - 2.0.93 ... 146
2.0.92 - What's new, December 12, 2023 ... 146
What's new in books and reporting - 2.0.92 ... 147
What's new in plans and applications - 2.0.92 ... 147
What's new in modeling - 2.0.92 ... 149
What's new in administration - 2.0.92 ... 150
Deprecation notices - 2.0.92 ... 152
2.0.91 - What's new, November 14, 2023 ... 153
What's new in books and reporting - 2.0.91 ... 153
What's new in plans and applications - 2.0.91 ... 157
What's new in modeling - 2.0.91 ... 160
What's new in administration - 2.0.91 ... 162
2.0.90 - What's new, September 19, 2023 ... 163
What's new in books and reporting - 2.0.90 ... 163
What's new in plans and applications - 2.0.90 ... 171
What's new in modeling - 2.0.90 ... 172
What's new in administration - 2.0.90 ... 174
Deprecation notices - 2.0.90 ... 175
2.0.89 - What's new, July 24, 2023 ... 176
What's new in books and reporting - 2.0.89 ... 176
What's new in plans and applications - 2.0.89 ... 180
What's new in modeling - 2.0.89 ... 181
Deprecation notices - 2.0.89 ... 183
2.0.88 - What's new, June 21, 2023 ... 183
What's new in books and reporting - 2.0.88 ... 183
What's new in plans and applications - 2.0.88 ... 187
What's new in modeling - 2.0.88 ... 188

What's new in administration - 2.0.88 ... 189
Deprecation notices - 2.0.88 ... 190
Known issues - 2.0.88 ... 190
2.0.87 - What's new, May 17, 2023 ... 191
What's new in books and reporting - 2.0.87 ... 191
What's new in plans and applications - 2.0.87 ... 199
What's new in modeling - 2.0.87 ... 200
What's new in administration - 2.0.87 ... 205
2.0.86 - What's new, April 18, 2023 ... 207
What's new in books and reporting - 2.0.86 ... 207
What's new in plans and applications - 2.0.86 ... 209
What's new in modeling - 2.0.86 ... 212
Upcoming change notifications - 2.0.86 ... 213
2.0.85 - What's new, March 22, 2023 ... 214
What's new in books and reporting - 2.0.85 ... 214
What's new in plans and applications - 2.0.85 ... 215
What's new in modeling - 2.0.85 ... 217
Known issues - 2.0.85 ... 220
2.0.84 - What's new, February 16, 2023 ... 220
What's new in books and reporting - 2.0.84 ... 221
What's new in plans and applications - 2.0.84 ... 222
What's new in modeling - 2.0.84 ... 224
Known issues ... 228
Deprecation notices - 2.0.84 ... 228
2.0.83 - What's new, January 19, 2023 ... 229
What's new in books and reporting - 2.0.83 ... 229
Known issues - 2.0.83 ... 234
Deprecation notices - 2.0.83 ... 234
2.0.82 - What's new, November 17, 2022 ... 234
What's new in books and reporting - 2.0.82 ... 235
What's new in plans and applications - 2.0.82 ... 236
What's new in modeling - 2.0.82 ... 237
Known issues - 2.0.82 ... 240
Deprecation notices - 2.0.82 ... 240
2.0.81 - What's new, October 20, 2022 ... 241
What's new in Planning Analytics Workspace Local - 2.0.81 ... 241
What's new in books and reporting - 2.0.81 ... 241
What's new in plans and applications - 2.0.81 ... 244
What's new in administration - 2.0.81 ... 245
What's new in modeling - 2.0.81 ... 246
Known issues - 2.0.81 ... 248
Deprecation notices - 2.0.81 ... 248
2.0.80 - What's new, September 22, 2022 ... 249
What's new in books and reporting - 2.0.80 ... 249
What's new in modeling - 2.0.80 ... 251
Deprecation notices - 2.0.80 ... 253
2.0.79 - What's new, August 25, 2022 ... 254
Planning Analytics Workspace Classic on Cloud no longer available as of 2.0.79 ... 255
Process editor available only on modeling workbenches ... 255
Track and report on plan progress with the plan Overview report ... 255
Send email reminders to contributors of open tasks ... 257
Set task dependencies for plan automation ... 258
Set a start date for your tasks ... 258
Use the new Features and Configuration tile to administer your Planning Analytics Workspace on Cloud environment ... 259
Configure an inactivity timeout for Planning Analytics on Cloud ... 259
Use the new interface to manage chores involved with a specific process ... 260
Select multiple variables in the process editor ... 261

Go to a specific line in the process editor....261
Apply formatting to visualization axis....262
Wrap text in data cells....263
Deprecation notice - chore editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.80....263
2.0.78 - What's new, July 13, 2022....263
Rules must now be created and maintained on a modeling workbench....264
New experience view now on modeling workbench....264
Approvals and Rejections in Plans....269
Reset password for Remote Desktop user accounts (Cloud only)....273
Updated menus for rows, columns, and data cells in views....274
Adjust page size in books....276
Enhanced data format feature....277
Open set editor for a benched dimension....278
Provide a direct path to a Planning Analytics Workspace folder....278
Update to Planning Analytics Workspace Local installation on Windows Server 2016....279
Enable email notifications in Planning Analytics Workspace Local....279
Deprecation notice - process editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.79....280
Deprecation notice for IBM Planning Analytics Workspace Classic experience....280
Known issues....281
2.0.77 - What's new, June 2, 2022....281
Upgrade explorations to the new experience....282
Changes to text properties for explorations....283
Select the sets that display in a dimension drop-down....285
Upload files to a Planning Analytics database with the action button....288
Manage files with File manager....289
Expand and collapse regions in the process and rules editor....290
Viewing the TM1 database version (admins only)....291
Auto-complete and popular search suggestions in the cognitive Learn pane....292
Updated menu options for visualization field slots....294
Hide hierarchy name to shorten visualization titles....295
Restore a book to the currently open tab....296
Deprecation notice - process editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.79....296
Deprecation notice for IBM Planning Analytics Workspace Classic experience....296
2.0.76 - What's new, May 5, 2022....297
Create and apply a custom theme for Planning Analytics Workspace....297
Send email notification when you open a plan step....299
Create control objects on the modeling workbench....300
Preview additional data source rows in the process editor....301
Sorting available members in the new experience set editor....301
Delete sets from the new experience set editor....302
Use the updated cell comment feature in the new experience cube viewer....303
Enhanced search and filter features....303
New properties for tree map visualizations....305
Customize the background color for data and header cells....306
Updated design for data spread options....307
Deprecation notice - process editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.79....308
Deprecation notice for IBM Planning Analytics Workspace Classic experience....309
2.0.75 - What's new, April 7, 2022....309
Displaying parents, children, or leaves for selected members....310
Update to predictive forecasting service....310
Adjust for outliers in your forecast....310
Changes to cube creation....311
Add variables in the Process editor....313
Search and filter on hierarchies....315

Deprecation notice - process editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.79 ... 316
Deprecation notice for IBM Planning Analytics Workspace Classic experience ... 316
2.0.74 - What's new, March 10, 2022 ... 317
Windows Server 2019 support (local only) ... 317
Add a view by dragging it from the data tree ... 317
Apply text styles to axis values in visualizations ... 318
Availability of color palettes for conditional formatting in Properties ... 319
Customize your explorations with new properties ... 320
Updated rules editor ... 321
Deprecation notice for IBM Planning Analytics Workspace Classic experience ... 322
2.0.73 - What's new, February 10, 2022 ... 322
Deprecation notice for IBM Planning Analytics Workspace Classic experience ... 323
Creation of modeling books no longer supported ... 323
Copy assets or folders to a shared or personal folder ... 323
Copy and paste images, pictures, web pages, and text ... 324
View all and View less links replaced by a menu in new experience view ... 324
Sort members, attributes, and properties in the new set editor ... 326
New process editor ... 326
Show agent label instead of IP address in Planning Analytics Administration (Local only) ... 327
Auto-start option for databases (Cloud only) ... 328
2.0.72 - What's new, December 20, 2021 ... 329
2.0.71 - What's new, December 14, 2021 ... 329
New experience view and set editor ... 330
New UI and functional improvements for Lifecycle Management ... 337
New header for plans and applications administration page ... 338
URLS in plan steps or application sections are now active links ... 338
Enable or disable consolidation formatting ... 339
Enhanced text formatting ... 339
Rename pins ... 341
2.0.70 - What's new, November 16, 2021 ... 341
Save a book as a template and set it as a default ... 342
Modeling workbench search improved to support control objects ... 342
New property added for item borders ... 343
Select multiple items by using mouse drag ... 343
Search on multiple words when monitoring thread activity in Planning Analytics Administration ... 344
Use custom compact numeric format to display values ... 344
Use enhanced workflow and capabilities when you create plans and applications ... 345
Submit a plan step for all groups to which you belong ... 345
Planning Analytics Workspace Local support for Red Hat Enterprise Linux 8 (Local only) ... 346
Deprecation notice - modeling books in Planning Analytics Workspace ... 346
2.0.69 - What's new, October 12, 2021 ... 346
Manage ODBC connections for multiple processes on the modeling workbench ... 346
New database-level search in the modeling workbench ... 347
Use the new Map visualization to view geographic data ... 348
Reset a book ... 349
Enhanced Scatter visualization ... 349
Rollback data entry button updated ... 350
2.0.68 - What's new, September 14, 2021 ... 350
Use Decision Optimization with Planning Analytics Workspace (Cloud only) ... 350
Analysts and modelers can create applications and plans ... 351
Dashboard grid size enhancements ... 352
Download a Welcome Kit directly from Planning Analytics Administration ... 353
2.0.67 - What's new, August 24, 2021 ... 353
Checkboxes available in the grid view ... 354
Pin widgets while in consumption mode ... 354
Assign a step or a section to everyone in plans/applications ... 355
2.0.66 - What's new, July 14, 2021 ... 355

The modeling workbench is fully integrated with the product.... 356
Full-page view of the modeling content.... 356
Apply proportional or relative proportional data spreading to a forecast.... 357
Hide zero/null data points in select visualizations.... 357
Confirm the Clear spreading shortcut command in the cube viewer.... 359
Include attribute headers when exporting a cube view.... 359
Use * and ? wildcards to search in the Set editor.... 359
2.0.65 - What's new, June 16, 2021.... 359
Use the modeling workbench to create and maintain your model (beta).... 360
See the updated view name when you save a view to the server .... 361
Abbreviated format to display large numbers.... 362
Assign multiple data items per slot in a visualization.... 362
Create calculations for named sets.... 363
**Minor changes to the procedures for creating plans and applications**.... 363
2.0.64 - What's new, May 18, 2021.... 364
**Use the Modeling workbench to create and maintain your model (beta)**.... 364
Forecasting improvements.... 365
Apply value formats in visualizations.... 365
Identical font size in equally sized widgets.... 366
2.0.63 - What's new, April 21, 2021.... 366
Design and usability improvements.... 366
Rename a TM1 database.... 369
Remove users in Planning Analytics Workspace.... 369
Use the new Box Plot visualization.... 369
**Edit MDX source for a cube view**.... 370
2.0.62 - What's new, March 17, 2021.... 370
Delete a TM1 database.... 371
**Seamlessly scroll through all objects in the data tree**.... 371
**Hide totals in a visualization with a new toolbar button**.... 371
**Retrieve data from multiple TM1 databases in a websheet (Planning Analytics Workspace local only)**.... 371
**Use scorecards to track objectives**.... 372
**Create asymmetric calculations on stacked rows or columns in an exploration**.... 372
**Synchronize parameters for an Action button that runs a TurboIntegrator process**.... 373
**Work with values formatted as Percentage in the Cube Viewer**.... 374
Known issue - Unexpected label in websheet Scatter Chart.... 374
2.0.61 - What's new, February 10, 2021.... 375
Create a TM1 database.... 375
Welcome Kit change for Planning Analytics on Cloud.... 375
Forecasting: preview nested dimension.... 375
**Sixty minute inactivity timeout for Planning Analytics Workspace on Cloud**.... 376
**View assets as tiles**.... 377
**Search at the node level in the data tree**.... 377
**Import dimension members from the data tree**.... 377
**Reset a sandbox**.... 378
2.0.60 - What's new, January 12, 2021.... 378
Forecasting: preview more than one row and forecast by using cube view data.... 378
Applications and Plans UI enhancements.... 380
New data tree Settings menu.... 381
Prefix negative values with a minus sign.... 382
Delete a folder for a removed user.... 382
Move assets or folders in a user's personal folder.... 383
2.0.59 - What's new, November 17, 2020.... 383
Create a global color palette.... 383
Support for iPad .... 384
Usability enhancements.... 385
Planning Analytics for Microsoft Excel: login or server access may fail in new experience .... 386
2.0.58 - What's new, October 21, 2020.... 386

Learn from the experts....386
Manage the new experience for your users....386
Improved look and feel....388
Applications and Plans....390
Administration page changes....391
Forecasting....392
Book and visualization improvements....393
Ask for help in the cognitive Learn pane....396
Considerations for upgrading to Planning Analytics Workspace new experience....397
Database administration enhancements....398
Planning Analytics for Microsoft Excel: login or server access may fail in new experience....398
2.0.57 - What's new, October 2, 2020....398
Learn from the experts....398
Improved look and feel....399
Applications and Plans....400
Administration page changes....401
Forecasting....402
Book and visualization improvements....403
Ask for help in the cognitive Learn pane....406
Considerations for upgrading to Planning Analytics Workspace new experience....407
2.0.55 - What's new, August 11, 2020....408
2.0.54 - What's new, July 15, 2020....408
Set your cookie preferences....408
Save a view to the TM1 database....410
Format numbers for an entire view....410
Suppress zeros for an entire view....410
Remove a user from all environments simultaneously (cloud only)....411

**Chapter 2. Get started with Planning Analytics Workspace....413**
Known issues....413
TurboIntegrator process connection to ODBC source fails in Planning Analytics on Red Hat Enterprise Linux 8 or 9....413
TurboIntegrator process comments starting with #Section or #Region should be modified before upgrading from Perspectives/Architect to Planning Analytics Workspace....413
Planning Analytics Workspace local/distributed fails to start when IPv6 is disabled on Linux using kernel boot parameter....414
User roles....415
Manage settings for components and features....416
Artificial intelligence capabilities in Planning Analytics Workspace....422
Find information....422
Planning Analytics Trial....424
Cookie preferences for Planning Analytics on cloud....425
Applicability of Planning Analytics Workspace documentation....426
Accessing Planning Analytics Workspace from Apple iPad....427
Supported languages....428

**Chapter 3. Work in books and views....429**
Books....429
Create a book....429
Edit mode....430
Save a book....430
Share a book....434
Resetting a book....434
Setting up the page in a book....435
Adjusting page size....435
Snapping objects to a grid and to each other....437
Change colors....437

Enable or disable item border in consumption mode....439
Customize tabs....441
Change the template on a tabbed book....442
Enable pinning in consumption mode....442
Set book properties....443
Add pictures, media, and web pages....443
Add cell values to the sheet as a cell view....445
Add shapes and text....446
Add pop-up text to books....448
Views....449
Add a new view to a book....449
Create views with Data explorer using natural language....449
Set properties for a view....451
Save a view....452
Change the display of data in a view....453
Setting view options....462
Change the format of data in a view....462
Analyze views with AI....474
Outlier analysis....474
Impact analysis....476
Exploration analysis....479
Visualizations in Planning Analytics Workspace....480
Create a visualization directly in a book....481
Visualizations available in Planning Analytics Workspace....483
Plotting members from the same dimension in certain charts....517
Change the format of data in a visualization....519
Use AI-generated insights to interpret a visualization....520
Matching color palettes on multiple visualizations....522
Setting visualization properties....523
Customize sets in dimension lists....524
Forecasting....526
Univariate and multivariate forecasts....527
Forecasting requirements....527
Forecasting options....532
Forecasting preview....539
Forecasting error messages....564
Forecasting on actuals....566
Forecasting models....567
Forecasting tutorial....570
Use Decision Optimization (Cloud only)....588
Decision Optimization prerequisites....589
Create a Decision Optimization job....589
View and run a Decision Optimization job....593
Synchronize objects in a book or sheet....593
Group synchronization....595
Setting synchronization for objects....601
Set synchronization for multiple widgets simultaneously....604
Data entry....606
Copy and paste....608
Comments....610
Apply data spreading....611
Hide rows and columns....612
Set word wrap options in row and column headers....613
Make asymmetric selections on rows and columns....613
Expand/collapse rows and columns asymmetrically or symmetrically....615
Lock row header column widths....617
Drill down....617
Drill through to detailed data....617

Sort rows and columns ... 618
Filter rows and columns based on cell value ... 618
Show and hide totals ... 620
Filter on top or bottom members ... 620
Suppress zeros ... 621
Expand levels ... 621
Display cell values as percentages ... 622
Refresh data ... 622
Set data refresh options for a view ... 623
Rollback data entry ... 624
Get cell explanation with AI ... 624
Open Planning Analytics for Microsoft Excel views ... 626
Conditional formats ... 626
Use the keyboard to work with conditional formatting ... 627
Calculations ... 629
Add a summary calculation to a view ... 629
Member calculations ... 630
Add a member calculation to a view ... 632
Add an asymmetric calculation to a view ... 632
Calculation formats ... 633
Rename a calculation ... 634
Delete a calculation ... 634
Find data ... 634
Find data with the tree ... 635
Sets ... 635
Create and edit a set ... 635
Editing a set (quick analysis) ... 647
Edit sets using the MDX editor ... 647
Add a set consolidation to a view ... 649
Action Buttons ... 652
Configuring the Action Button to run an action ... 653
Set process parameters for a button ... 655
Upload files to a database with the action button ... 657
Websheets ... 659
Save a websheet to the database ... 660
Open a websheet on the active tab when you save a multi-tab websheet ... 661
Explore scenarios with sandboxes ... 661
Compare sandboxes ... 662
Use the Planning Analytics AI assistant to interact with books using natural language ... 663
Snap commands ... 669
Access your data and other objects quickly ... 671
Pin items ... 671
Access recently used items ... 672
Bookmark items ... 672
Burst export to PDF or PowerPoint ... 673
Export to Excel ... 673
Export data from a cube or view ... 676
Export a dimension or hierarchy ... 677
How to parameterize an MDX query in IBM Planning Analytics Workspace ... 678

**Chapter 4. Use applications and plans to organize work ... 681**

Applications ... 681
Create an application ... 682
Manage an application ... 686
Pass context between books in an application ... 687
Convert an application to a plan ... 689
Open and use an application ... 690

Plans....691
Create a plan.... 691
Email notification in plans....698
Pass context between books in a plan.... 699
Track and report on plan progress with the plan Overview report .... 701
Track tasks with the My tasks tab....703
Contribute to a plan....704
Manage a plan.... 706
Convert a plan to an application.... 710
Automate plan actions with workflow processes.... 711

**Chapter 5. Explore scorecards....713**
Scorecards.... 713
Impact diagram....713
Strategy map.... 713
Custom diagram.... 714
Metrics cubes....714
Traffic light status indicator.... 715
Trend indicator.... 715

**Chapter 6. Model in Planning Analytics Workspace.... 717**
The modeler role.... 718
Steps to building a model....718
Use the modeling workbench to create and manage your model....719
Working in the modeling workbench.... 720
Create a modeling workbench....727
Work with assets on the modeling workbench .... 727
Organize editors on the workbench....729
Open editors in a separate window.... 730
Using database-level search in a modeling workbench.... 730
Workbench preferences....734
Using the File manager.... 734
Using the Database explorer....737
Copy menu on the modelling workbench....738
Change the security of a websheet....738
Make a websheet public or private....739
Create shortcuts to TM1 objects in the modelling workbench....739
Dimensions.... 740
Using the dimension editor....741
Create a dimension in Planning Analytics Workspace.... 744
Format dimension members in Planning Analytics Workspace.... 750
Add members to a dimension or hierarchy in Planning Analytics Workspace.... 760
Time dimensions in Planning Analytics Workspace....762
Reserve and release a dimension in Planning Analytics Workspace....764
Export a dimension or hierarchy from a modeling workbench....765
Lock and unlock a dimension in Planning Analytics Workspace.... 766
Lock and unlock an element in IBM Planning Analytics Workspace.... 766
Manage dimension properties.... 767
Manage dimension attributes.... 768
Hierarchies.... 769
Member attributes.... 771
Pick lists....772
Dimension editor search and locate in hierarchy....777
Identify which cubes include a specific dimension.... 777
Cubes.... 778
Create a cube.... 780
Recreate a cube from an existing cube.... 781

Export data from a cube or view....782
Importing data into a cube....783
Cube import file format....786
Optimize the order of dimensions in a cube....788
Lock and unlock a cube....790
Unload and reload a cube from memory....790
Delete a cube or a view....790
Manage cube properties....791
Manage cube attributes....794
Rules....794
Create and edit rules....795
Use the rules editor in Planning Analytics Workspace....795
Delete rules....797
Trace cell values....797
Check feeders....798
Trace feeders....799
TurboIntegrator processes....800
Create and edit processes....800
Insert a code snippet into a process....810
Use the process editor....820
Run a process....822
Edit chores that involve a specific process....824
Remove a process from multiple chores....824
Debug a process....824
Delete a process....827
Zero suppression in a TurboIntegrator process affects member ordering in a view....827
Manage process attributes....828
TurboIntegrator chores....828
Create and edit chores....829
Execute a chore on demand....830
Enable or disable a chore schedule....831
Delete a chore....831
Manage chore attributes....831
Running a chore at database startup....831
Baseline forecasts....832
Create the baseline forecast asset and set forecast details....832
Define a baseline forecast timeframe....833
Specify the scope of a baseline forecast....834
Specify the variables of a multivariate baseline forecast....834
Define advanced baseline forecast settings....835
View the baseline forecast summary and activate the baseline....836
Edit a baseline forecast....838
Run a baseline forecast....838
Schedule a baseline forecast....839
Control objects....842
Create a control dimension....842
Create a control cube....843
Create a control process....843
Configure drill-through to reveal detailed data....843
Create a drill rule....844
Edit a drill rule....845
Delete a drill rule....846
Create a drill process....846
Edit a drill process....849
Delete a drill process....849
Object security....849
Create a user group....849
Add users to user groups....850

Secure dimension members....851
Secure cubes, dimensions, processes, and chores....853
Manage security for dimension hierarchies....854
Secure cells....855
Delete a user group....858
Localize a model....858
Localize cubes, dimensions, processes, or chores....858
Localize dimension members....859
Localize set names....860
Localize view names....861
Manage Planning Analytics Workspace database log files....862
Server log....863
Audit log....864
Transaction log....864
Roll back transactions in the transaction log....865
Manage database threads on the modeling workbench....865
Start and stop performance monitoring from the modeling workbench (TM1 Database 11 only)....866
Use Git repository integration to manage Planning Analytics database objects....867
Add database objects to a Git repository....868
Update your model with database objects from a Git repository....870
Reset a Git repository connection....871

**Chapter 7. Administer IBM Planning Analytics Workspace....873**
Download a Welcome Kit from Planning Analytics Administration....873
Administer users and groups....873
Administer users on cloud....874
Manage users and groups on a Planning Analytics database (Cloud only)....887
Administer local users....890
Create users in a Planning Analytics Local database....900
Administer groups....901
Export users....902
Change a user's role....903
Change a user's environment (Cloud only)....904
Copy and deploy assets with Lifecycle Management in Planning Analytics Workspace....905
Prerequisites....905
Limitations....906
Get started....906
Create a snapshot....907
Deploy a snapshot....908
Export a snapshot....909
Import a snapshot....910
Rename or delete a snapshot....910
View a snapshot log....910
Manage features....911
Disable select data spreading methods on Planning Analytics Workspace Local....911
Configure the Planning Analytics Workspace environment....912
Configure an inactivity timeout for Planning Analytics on Cloud....912
Create and edit multiple Planning Analytics Local environments (Local only)....913
Monitor and administer databases....914
Connection timeouts in Planning Analytics Workspace on Cloud....915
What are the differences between Planning Analytics Administration on premises and on cloud?....915
Configure system monitoring....916
Install and configure the Planning Analytics Administration agent (local only)....919
Monitor and manage database threads....927
Create a database (Cloud only)....929
Rename a database....931

Delete a database ... 932
Identify the database used for authentication on Planning Analytics Administration Local (Local only) ... 933
Configure databases ... 934
Start and stop databases ... 940
Download database log files ... 941
Save data for a selected database ... 941
Refresh security for a selected database ... 941
View failed database requests ... 941
Download additional components ... 943
Customize the interface for your company ... 944
Upload custom fonts ... 944
Create a global color palette ... 945
Create a custom theme ... 947
Manage integrations ... 952
Configure integration with IBM watsonx.ai to support Decision Optimization ... 952
Connect the Planning Analytics AI assistant to a custom instance of IBM watsonx Orchestrate ... 954
Additional administration capabilities for IBM Planning Analytics as a Service ... 955
Review your Service Instance Quota for IBM Planning Analytics as a Service ... 955
Manage your IBM Planning Analytics as a Service environments ... 956
Edit an environment name ... 956
Manage databases on Planning Analytics as a Service ... 957
Set permissions for assets ... 961
Set permissions for assets in Planning Analytics Workspace ... 962
Set permissions for a book in Planning Analytics Workspace Classic ... 962
Set permissions on the Shared folder and subfolders ... 963
Set permissions on the Shared folder and subfolders in Planning Analytics Workspace ... 963
Set permissions on the Shared folder and subfolders in Planning Analytics Workspace Classic ... 965
Limit access to the Set Editor ... 966
Limit access to the Set Editor Planning Analytics Workspace ... 966
Unload and reload a cube from memory ... 966
Access the S:/ drive and files in the remote server ... 967

**Appendix A. References ... 971**
Rules functions ... 971
Arithmetic operators in Planning Analytics rules ... 971
Comparison operators in Planning Analytics rules ... 971
Logical operators in Planning Analytics rules ... 971
Attribute rules functions ... 972
Consolidation calculation rules functions ... 977
Cube data rules functions ... 985
Date and time rules functions ... 990
Dimension Information Rules Functions ... 998
Element Information Rules Functions ... 1002
Financial Rules Functions ... 1015
Hierarchy Rules Functions ... 1017
Logical Rules Functions ... 1019
Mathematical Rules Functions ... 1020
Text Rules Functions ... 1027
Miscellaneous Rules Functions ... 1035
TurboIntegrator Functions ... 1036
TurboIntegrator reserved words ... 1037
ASCII and Text TurboIntegrator Functions ... 1037
Attribute Manipulation TurboIntegrator Functions ... 1046
Chore Management TurboIntegrator Functions ... 1097
Cube Manipulation TurboIntegrator Functions ... 1099
Data Reservation TurboIntegrator Functions ... 1111

Date and Time TurboIntegrator Functions....1115
Dimension Manipulation TurboIntegrator Functions....1118
Hierarchy Manipulation TurboIntegrator Functions....1131
ODBC TurboIntegrator Functions....1146
Process Control TurboIntegrator Functions....1149
Rules Management TurboIntegrator Functions....1159
Sandbox Functions....1165
Security TurboIntegrator Functions....1175
Server Manipulation TurboIntegrator Functions....1189
Subset Manipulation TurboIntegrator Functions....1194
View Manipulation TurboIntegrator Functions....1214
Miscellaneous TurboIntegrator Functions....1230
TurboIntegrator Variables....1237
TurboIntegrator Local Variables....1237
TurboIntegrator Global Variables....1245
Implicit Global Variables....1246
TurboIntegrator User Variables....1248
MDX Function Support....1248
Support for Microsoft-defined MDX Expressions and Functions....1248
TM1 specific MDX functions....1250
TM1 specific MDX expressions....1253
Data spread keyboard shortcuts....1254
Proportional data spreading method....1256
Equal data spreading method....1257
Repeat data spreading method....1257
Clear data spreading method....1258
Percent change data spreading method....1259
Repeat leaves data spreading method....1259
Equal leaves data spreading method....1260
Straight line data spreading method....1261
Growth percent data spreading method....1261
Relative proportional data spreading method....1262
Relative percent adjustment data spreading method....1262
Exploration navigation shortcuts....1263
Naming conventions....1264
Map reference info....1267
Supported languages for maps....1267
Troubleshooting Planning Analytics Workspace....1268
Setting the TM1 Web session timeout....1268
Slow scrolling in Google Chrome browser....1269
Help error: Unable to get resource file....1269
Accessibility....1269

**Notices....1271**

# Introduction

IBM® Planning Analytics Workspace is a web-based interface for IBM Planning Analytics. It provides an interface to TM1® data, with ways to plan, create, and analyze your content.

Planning Analytics Workspace helps you focus on the things that matter to your business. Using Planning Analytics Workspace, you can identify and understand patterns and relationships in data. You can use this information to understand why things happen, and predict what might happen. Planning Analytics Workspace opens up the world of advanced analytics to all business users.

## Accessibility features

Accessibility features help users who have a physical disability, such as restricted mobility or limited vision, to use information technology products.

IBM HTML documentation has accessibility features. PDF documents are supplemental and include no extra accessibility features. For accessible documentation, see IBM Knowledge Center.

## Forward-looking statements

This documentation describes the current functionality of the product. References to items that are not currently available may be included. No implication of any future availability should be inferred. Any such references are not a commitment, promise, or legal obligation to deliver any material, code, or functionality. The development, release, and timing of features or functionality remain at the sole discretion of IBM.

## Samples disclaimer

The Sample Outdoors Company, Great Outdoors Company, GO Sales, any variation of the Sample Outdoors or Great Outdoors names, and Planning Sample depict fictitious business operations with sample data used to develop sample applications for IBM and IBM customers. These fictitious records include sample data for sales transactions, product distribution, finance, and human resources. Any resemblance to actual names, addresses, contact numbers, or transaction values is coincidental. Other sample files may contain fictional data manually or machine generated, factual data compiled from academic or public sources, or data used with permission of the copyright holder, for use as sample data to develop sample applications. Product names referenced may be the trademarks of their respective owners. Unauthorized duplication is prohibited.

© Copyright IBM Corp. 2009, 2025

# Chapter 1. What's new in Planning Analytics Workspace

Find out what's new or changed in the most recent release of IBM Planning Analytics Workspace.The new features described here always reflect the full capabilities of Planning Analytics Workspace on cloud.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

## 2.0.107 - What's new, September 24, 2025

Learn about new features and known issues in version 2.0.107 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.



**Attention:** The 2.0.107 release is the last release of Planning Analytics Workspace to use a 2.0 version number. The next update for Planning Analytics Workspace will be 2.1.15.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

### What's new in books and reporting - 2.0.107

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.107.

#### Set permissions across multiple assets

Set permissions on multiple assets at once in both the **Reports and Analysis** and the **Data and Models** areas.

You can select multiple items by selecting the corresponding checkboxes, then use the new **Set permissions** option in the multi-select toolbar to apply the changes to all items at once.

When a folder is included in your selection, a new **Apply permission changes recursively** option appears (enabled by default) to apply the permission updates to all nested content. This option also appears when copying or moving assets, which allows you to define permissions before the action begins. Previously, permissions were set in a separate dialog after you started copying or moving the asset.

Additionally, the **Edit only** permission is renamed to **View and edit** to better reflect its function.

**Note:** These changes do not apply to Plans and Applications.

#### Customize notifications for processes run from action buttons

Specify custom notification messages when processes exit early when run from an action button.

You can use action buttons in Workspace books to run TurboIntegrator processes. Previously, if a process that was run from an action button returned a quit status, the action button treated it as a successful run and displayed the success message. With this feature, you can now configure distinct messages for both successful and quit outcomes, which gives users clearer feedback when a process exits early.

© Copyright IBM Corp. 2009, 2025

Customize notifications from the **Customize notifications** section in the **Action button** configuration page:

1. In **Edit** mode, select the action button that you want to configure.
2. Under the **Properties** > **Configure** menu, browse to the **Customize notifications** section.
3. Specify the messages for the **Success** and **Quit** statuses.

### Content navigation performance improvements

Content navigation is now faster in the **Reports and Analysis**, **Data and Models**, and **Plans and Applications** areas.

The performance improvements also result in faster load times when you browse folders (especially folders with many assets) and when you copy or move content.

To support these improvements, Planning Analytics Workspace now uses end-string truncation for long asset names and displays the full name in a tooltip. You can re-enable mid-string truncation by selecting the **Mid truncation** field in the **Settings** icon from each content navigation page, but doing so might reduce performance when you are selecting assets.

## What's new in plans and applications - 2.0.107

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.107.

### Assign ownership and track progress with data accountability

Use data accountability to assign specific slices of cube data to Planning Analytics Workspace groups in your plan.

This makes contributors and reviewers accountable for submitting changes, analyzing data, and approving or rejecting submissions for their assigned members in a TM1 hierarchy.

You configure accountability with a validated subset or hierarchy from your TM1 model. Each member becomes a node with a workflow state (available, reserved, pending, complete, or in progress) tracked in a state cube. As users interact with the plan, node states update automatically, and changes cascade to ancestor and descendant nodes to reflect progress.

Data accountability also introduces task history, notifications, and TM1 objects that support tracking and automation. Contributor and reviewer actions, like taking ownership, submitting, approving, or rejecting, are logged and trigger updates to the plan state. This structured workflow experience helps you manage responsibility across your planning process and mirrors key aspects of TM1 Applications.

Data accountability is disabled by default. To enable data accountability, go to the **Planning Analytics Administration** page, click the **Features** tab, then enable the **Data accountability** field.

### Automate plan actions with workflow processes

Use workflow processes to trigger TurboIntegrator actions based on user activity in Planning Analytics Workspace plans.

You can configure TurboIntegrator processes to run when tasks are activated or deactivated, when contributors take ownership or submit changes, and when reviewers approve or reject submissions.

Workflow processes pass plan-specific details (such as the plan ID, task ID, and member from the data accountability hierarchy) into the TM1 process. You can also define static values for parameters, making it easier to reuse processes across multiple events. Use workflow processes to perform tasks such as assigning security, managing TM1 groups, and locking or unlocking data slices by using the TM1 security overlay feature.

Processes run under the credentials of the user who triggered the event and execute after the plan state is updated. To ensure successful execution, users must have appropriate access in the TM1 Database.

Workflow processes are disabled by default. To enable workflow processes, go to the **Planning Analytics Administration** page, click the **Features** tab, then enable the **Workflow processes** field.

# What's new in modeling - 2.0.107

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.107.

## Open process logs in a dedicated workbench tab

View and debug process logs in a separate tab while you are working with the process.

You can now open process logs as a separate tab in the modeling workbench. Previously, logs were only available in a modal window. With this enhancement, you can open logs in multiple tabs and monitor them in real time while you are editing or running processes, which makes process development and troubleshooting more convenient.

Open process logs as a separate tab from the **Processes** node on the **Databases** tree in a modeling workbench:

1. On a modeling workbench, expand the database where you want to examine process log files.
2. Right-click the **Processes** node, then click **Process logs**.

   The logs open in a new workbench tab.

## Hierarchy editor "expand all" option no longer reveals hidden members

From the hierarchy editor, the **Expand all** button no longer reveals members that were previously hidden by using the **Keep** or **Hide** options. This change makes sure that hidden members remain hidden even when expanding all consolidated members.

You can still expand or collapse individual members using the icons next to their names without affecting hidden members. This update helps preserve your customized view when working with large or complex hierarchies.

# What's new in administration - 2.0.107

The following new and changed administration features are available in Planning Analytics Workspace 2.0.107.

## Capabilities access setting is limited in Planning Analytics Administration

The **Capabilities access** setting is no longer available in Planning Analytics Administration for users who are not members of the environment.

When managing user environments in Planning Analytics on Cloud, the **Capabilities access** toggle is disabled for environments for which you are not a member. The toggle still appears, but is greyed out for users that are not members of the environment.

Similarly, in the **Manage environments** window from the **Users and Groups** page for Planning Analytics Administration, the **Capabilities** toggles are also disabled and greyed out for users that are not members of the environment.

To modify capabilities in a remote environment, first add the logged-in user to that environment, then enable the capability from within the environment. This change helps ensure that access permissions align with the specific environment.

### View the database creator in Planning Analytics Administration (Planning Analytics as a Service only)

When you are working with TM1 Database 12 in Planning Analytics as a Service, you can now see the name of the user who created the database.

This information appears in the database properties and helps administrators track ownership and manage environments.

The creator is identified as the Planning Analytics Workspace administrator who initialized the database, and this information is available directly in Planning Analytics Administration.

## 2.0.106 - What's new, July 29, 2025

Learn about new features and known issues in version 2.0.106 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.106

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.106.

### Synchronize sets to action icons in Planning Analytics Workspace books

Pass dynamic sets to TurboIntegrator processes by using action icons.

You can now synchronize sets, in addition to individual members, to action icons in Planning Analytics Workspace books. You can synchronize sets to pass a set of members as an MDX expression to a string parameter in a TurboIntegrator process.

To use this feature, configure an action icon to run a process with one or more string parameters. In the parameter settings, select **Synchronize set**, then select the synchronization group, dimension, and hierarchy. At run time, the string parameter receives an MDX expression such as the following expression:

```
{TM1SubsetToSet([dimension].[hierarchy], "subset_name", "public")}
```

If a selection of members is used, the same MDX statement is used with a session set ID. For example,

```
{TM1SubsetToSet([dimension].[hierarchy], "sessionSetID")}
```

These MDX expressions can be used in your TurboIntegrator code with functions that accept an MDX statement as a parameter, such as `SubsetCreateByMDX`.

The updated action icon configuration interface replaces the previous synchronization group list with **Synchronize member** and **Synchronize set** options.

This feature is available to Planning Analytics Workspace book authors.

## Edit calculations in the cube viewer

Update calculated members without re-creating them from scratch.

You can now edit existing calculations directly in the cube viewer. When you right-click a calculated member, a new **Edit calculation** option appears in the menu. Selecting this option opens the appropriate calculation dialog, and the current formula is already entered into the field. You can modify the calculation and rename it if necessary.

This feature supports calculations that you already created by using the cube viewer's built-in calculation interfaces. You cannot use this option to edit calculations that you created by manually editing the MDX statement.

## Use parameterized MDX queries in Planning Analytics Workspace books

Create dynamic, flexible views by using advanced MDX parameterization.

You can now use parameterized MDX queries in Planning Analytics Workspace books to build views that go beyond the limitations of standard widget synchronization.

To use this feature, open the cube viewer and click the **Edit MDX** icon in the toolbar. You can then define parameters and reference them within your MDX query. This feature is available to all users who have access to the MDX editor.

**Note:** When you are using parameterized MDX, some view features (such as moving hierarchies between axes) are not available. This is an existing limitation from the original parameterization feature.

## Show distinct ancestors in the Set editor

Control how parent and ancestor members appear when you expand selections.

A new **Distinct ancestors** setting in the Set Editor determines how parent and ancestor members are displayed when you are using the **Expand to** or **Keep with** options. When you de-select this setting, multiple instances of the same ancestor might appear if a selected member has multiple parents. When you select this setting, only one instance of each ancestor is displayed.

This setting works by automatically applying the **`DISTINCT`** MDX function to the query.

To enable or disable this option, open the Set editor, click the **Settings** icon, then click the **Distinct ancestors** field.

## Use up to three external variables in multivariate forecasts

Improve forecast accuracy by including more influencing factors.

Multivariate forecasting now supports up to three external variables so you can model more complex relationships in your data. Previously, you could specify up to two external variables in multivariate forecasts. These variables help the forecast engine understand how different time-dependent factors interact with your historical data, which leads to more accurate and insightful predictions.

To use this feature, configure your multivariate forecast as usual from the **Set up** tab and select up to three variables that are correlated with your target data. The addition of each variable is correlated to see the impact on the accuracy of each forecasting member. Additional variables might enhance forecast precision but might also increase calculation time.

## Get insights generated for your forecasts

Understand your forecast results with natural language summaries powered by watsonx.

Forecast insights provide AI-generated explanations that describe the forecasted values and model accuracy for univariate forecasts. These summaries help you interpret results more easily and make informed decisions based on the forecast output.

To use this feature, set up a univariate forecast and click **Preview**. The insights display with the forecast results.

**Note:** Forecast insights are currently available only for univariate forecasts and do not support on-demand multivariate forecasting.

### Export multivariate forecast values to a CSV file

You can now export forecast values from the preview chart to a CSV file when you are viewing both univariate and multivariate forecast previews.

Previously, you could export forecast values for univariate values only. Now, you can export forecast values for both univariate and multivariate forecasts. This gives you the opportunity to analyze and work with projected forecast data before submitting it to the cube (database), helping you make more informed decisions.

To export forecast values from the preview chart, click **Download data as CSV**.

The resulting CSV files for both univariate and multivariate forecasts contain data exported into the following columns:

1. Name of the tuple or the selected row.
2. Forecast periods from the time dimension.
3. Forecast slice, which consists of forecast values.

   **Note:** In univariate forecasts, this also consists of high values and low values.
4. Actual values for each of the slices and the corresponding forecast time period.

### Use Data explorer with natural language commands

Use the Planning Analytics AI assistant to generate cube views with a simple prompt.

You can now trigger the Data explorer AI feature directly from the Planning Analytics AI assistant. You can now describe the data that you are looking for in natural language, and the Planning Analytics AI assistant opens the Data explorer tearsheet, selects the appropriate server and cube, then generates a cube view.

To use this feature, type a command like one of the following phrases:

- `Explore my data on ‘my_server’ with this generic query...`
- `Explore my data on ‘this_cube’ on ‘my_server’ with this specific query...`

If you do not specify a server, the Planning Analytics AI assistant uses the current search context. If only one database is available, the Planning Analytics AI assistant proceeds automatically. If multiple databases are found, you are prompted to choose one. If no context is available, a drop-down list of all databases is shown.

This feature is available for users who have access to both the Planning Analytics AI assistant and the Data explorer.

## What's new in plans and applications - 2.0.106

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.106.

### Keep the Contribution panel open or closed based on your preference

Maintain your preferred **Contribution** panel state (open or closed) across sessions in the **Plans** and **Applications** pages.

The **Contribution** panel which appears in the **Plans** and **Applications** pages helps you navigate between sections, tasks, assets, and announcements. You can open or close this panel at any time by using the **Contribution** panel button even if you are not in Edit mode.

Now, when you close the panel, the interface retains this state. The next time you log in and open any plan or application, the panel remains closed until you choose to open it again. This preference is saved at the user level and applies across all plans and applications.

# What's new in modeling - 2.0.106

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.106.

## Save export options with your cube view

Reuse your preferred CSV export settings without re-configuring them each time.

When exporting data from a cube view to CSV, you can choose to include or exclude zeros, consolidations, and rule-derived values. You can also filter the export to specific numeric or string ranges. Now, you can optionally save these export settings with the view, so that your preferences are already applied the next time you export from the same view.

If you change any export options during the export process, you will be prompted to save them to the view. Only users with the **Administrator** role or who belong to a group with Administrator-level security on a cube can update public views. All users with read access can update their own private views.

**Note:** These export settings also affect TurboIntegrator processes that use the view as a data source if you are using TM1 version 11.8.31 or later. The settings correspond to the **`ViewExtractSkipCalcsSet`**, **`ViewExtractSkipRuleValuesSet`**, and **`ViewExtractSkipZeroesSet`** functions.

## Open a duplicate Drive explorer tab in the modeling workbench (Cloud only)

Work more efficiently by opening a second Drive explorer tab to the same directory in the modeling workbench.

You can now open a duplicate Drive explorer tab in the modeling workbench by clicking the new **Duplicate** icon in the Drive explorer, which is located with the other action icons for the tab. The duplicate tab opens to the current directory in the original instance of the Drive explorer and reuses your existing credentials. You can use this feature to browse or manage files in parallel without losing your place. However, you cannot copy or paste files between the different Drive explorer tabs.

## Open duplicate Cube viewer and File manager tabs in the modeling workbench

Work more efficiently by opening multiple tabs for the same object in the modeling workbench.

You can now open duplicate tabs for the Cube viewer and the File manager in the modeling workbench. To duplicate a tab, click the new **Duplicate** icon in that tab, which is located with the other action icons. The duplicate tabs point to the same TM1 object as the original tab. You can use this feature to view or edit the same object in parallel without browsing away from your original place in the object. This feature does not duplicate the underlying TM1 object, rather, it opens another view of the object.

## View and manage control objects in the Database explorer

Perform bulk actions on control objects directly from the Database explorer.

You can now view and interact with control objects (that is, cubes, dimensions, and processes that begin with "}") in the Database explorer. This enhancement expands the range of objects that are available for multi-selection and bulk actions, such as deleting multiple items at once.

You can access this feature from the modeling workbench if you have the **Modeler** role. It builds on the existing Database explorer functionality by including control objects alongside standard TM1 objects.

You must have the **Modeler** role to use this feature.

## Set a default member for a hierarchy

Control how data queries select members by specifying a default member in your hierarchy.

You can now set a default member for any hierarchy in the dimension editor. The default member is automatically used when no specific member is referenced in, for example, MDX queries that don’t

explicitly include the hierarchy. Selecting a default member helps you to obtain consistent query results and simplifies report development.

To set the default member, open the hierarchy in the dimension editor, right-click the member, then select **Set as default**. A star icon indicates the selected default member.

## Run TurboIntegrator processes from Application shortcuts

Run processes directly from Application shortcuts in the modeling workbench.

You can now run a TurboIntegrator process from its shortcut in the Applications pane. This enhancement adds a **Run** option to the menu of process shortcuts, so you can trigger processes without browsing through the database tree or opening the process editor.

To use this feature, create a shortcut to a TurboIntegrator process. Then, right-click the shortcut and select **Run**. This feature is available to users with the **Modeler** role and requires TM1 Database 11 (version 11.8.29 or later), or TM1 Database 12.

**Note:** Note: Process shortcuts are only available in the modeling workbench and do not appear in Planning Analytics Workspace books.

## Open all shortcuts in an Applications folder at the same time

Start multiple TM1 object editors from a single Applications folder.

You can now open all shortcuts within an Applications folder in the modeling workbench with a single action. Each shortcut opens as a separate tab, so you can work with multiple TM1 objects more efficiently.

To use this feature, right-click an Applications folder in the TM1 Database tree and select **Open all shortcuts** from the menu. This selection opens only shortcuts in the immediate folder, while shortcuts in subfolders are not included. You cannot open some object types, such as websheets, in the modeling workbench. This feature is available to users with the **Modeler** role and requires TM1 Database 11 (version 11.8.29 or later), or TM1 Database 12.

## Search Applications folders in the TM1 Database

Find objects in an Applications folder by using the search box.

You can now search for objects within an Applications folder in the TM1 Database tree. Previously, search was limited to cubes, dimensions, views, and other core objects. This enhancement makes it easier to locate shortcuts and folders when you work with large Applications structures.

To use this feature, expand an Applications node in the TM1 Database tree. If the folder contains 10 or more items, a search box appears at the beginning of the list. This feature is available in both the modeling workbench and the Planning Analytics Workspace books.

## Allow users to select different slices for statistical details in baseline forecasts

An administrator or modeler can now set an option that enables users to save statistical details in different slices for baseline forecasts.

When users run a baseline forecast, they can now store statistical details in different slices, overriding the default selection. Use different slices to manage multiple user that run concurrent forecasts by keeping each user's statistical output separate, making it easier to review and refine forecasts over time.

Previously, you can choose to store statistical details by enabling the **Save statistical details ...** option at the **Summary** page for the baseline forecast. You can now enable the **Allow override** option, then a user can select a preferred slice to store the statistical details. When your users run the baseline forecast, they can change the slice before they run the forecast, which is similar to the option that enables users to override default context and forecast members.

# What's new in administration - 2.0.106

The following new and changed administration features are available in Planning Analytics Workspace 2.0.106.

## View the TM1 Server version number in Planning Analytics Administration

You can now see the version number of your Planning Analytics (TM1 Server) installation directly in Planning Analytics Administration.

This feature helps you quickly verify which version is running without needing to access the server directly. To view the version number, go to the **Agents** tab in the **Databases** page from the Planning Analytics Administration interface. This feature is available only to users with the **Administrator** role.

**Note:** This feature requires TM1 Server version 2.0.9.22.

## Connect the Planning Analytics AI assistant to a custom IBM watsonx Orchestrate instance

Use your own watsonx Orchestrate configuration to extend the capabilities of the Planning Analytics AI assistant.

You can now configure the Planning Analytics AI assistant to connect to a custom watsonx Orchestrate instance instead of the default instance. Using your own watsonx Orchestrate instance extends the Planning Analytics AI assistant to start your own skills and intents (such as sending emails or triggering workflows) directly from the Planning Analytics AI assistant chat.

After users enable a custom configuration, they can interact with the Planning Analytics AI assistant as usual. For example, typing "`send an email`" triggers the corresponding skill from the connected custom watsonx Orchestrate instance if it is properly configured.

Enable this feature from the **Planning Analytics Administration** page:

1. Go to the **Planning Analytics Administration** page.
2. Click **Integrations** and select **IBM watsonx Orchestrate**.
3. Click **Custom instance** and in the **Configuration value** field, paste your custom configuration JSON string.
4. Click **Test** to verify the connection.
5. If your connection is successful, click **Apply** to activate the configuration.

# Known issues - 2.0.106

The following are known issues in Planning Analytics Workspace 2.0.106. These issues will be resolved in a future release.

## Forecast summary displays error message if watsonx Orchestrate is not enabled (local only)

If you are using Planning Analytics Workspace Local and you did not enable watsonx Orchestrate because you do not have the Planning Analytics AI assistant, the **Forecast summary** tab in the forecasting preview displays the following error message: `Network response was not ok`.

To verify that watsonx Orchestrate is not enabled, check the **Integrations** page:

1. Go to the **Planning Analytics Administration** page.
2. Click **Integrations** and select **IBM watsonx Orchestrate**.
3. watsonx Orchestrate is not enabled if the current setting is **Off**.

If you create a forecast and click **Preview**, the **Forecast summary** tab displays the error message.

### Users can have different email addresses in different environments

If you are using multiple Planning Analytics Workspace Local environments, your account can have different email addresses listed for the different environments.

## 2.0.105 - What's new, June 25, 2025

Learn about new features and known issues in version 2.0.105 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.105

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.105.

### Display Planning Analytics component versions

The **About Planning Analytics** dialog in Planning Analytics Workspace now includes detailed version information for key components.

The detail version information that is now displayed includes the following:

- The specific build number for Planning Analytics Workspace
- The version of the Planning Analytics Spreadsheet Services (PASS)

This feature provides greater transparency and helps users and administrators quickly verify the versions of deployed components.

Access this information as follows:

1. Click your user name on the Planning Analytics Workspace title bar.
2. Click **About Planning Analytics**.
3. Click the **Runtime versions** link in the dialog to view detailed version information.

**Note:** Planning Analytics local installations might not show PASS version details if you did not deploy PASS in your environment.

### Configure empty pick list value

You can now customize or remove the default empty value label in pick list drop-down lists within cube views.

Previously, pick list drop-down lists in the cube viewer cells always included a default **--None--** option that allowed users to clear the current cell value. Book authors can now configure the label for this empty value or remove it entirely from the drop-down field.

Authors now have more control over the following characteristics of pick lists:

- Replace the default **--None--** label with a custom string.
- Remove the empty value option from the drop-down list entirely.

Even when the empty value is removed, users can still clear a pick list cell using the **Delete** key or a clear spread.

You can access this setting in the **Format manager** for each cube view widget in a Planning Analytics Workspace book:

1. From the cube view widget, click the **Format manager** icon to open the Format manager.
2. Click **Data area**.
3. Navigate to the **Data entry** section.
4. Configure the **Picklist empty value** field:
   - To enable the pick list empty option, select the **Picklist empty value** checkbox and specify the text string that users click to clear the current cell value. The default text string is `--None--`.
   - To disable the pick list empty option, clear the **Picklist empty value** checkbox.
5. Click **Apply**.

### Drill views display full toolbar and Set editor

Drill views in the Planning Analytics Workspace cube viewer now show the full toolbar and Set editor, significantly improving usability and consistency with source views.

Previously, drill target views lacked key functionality such as zero suppression, export, save, sandbox support, and access to the Set editor. With this feature, drill views now include the same toolbar features that are available in the source view.

Toolbar behavior is inherited from the drill source view. For example, if the sandbox feature or Set editor is disabled in the source view, it will also be disabled in the drill target view. This ensures a consistent and controlled user experience across both views.

To use this feature, navigate to a drill view from a cube viewer widget in a Planning Analytics Workspace book or from the cube viewer in the modeling workbench.

### Updates to OpenShift storage configuration (local only)

Planning Analytics Workspace now includes improved storage configuration handling for deployments on OpenShift®.

These updates simplify and streamline the configuration process by automatically updating the following environment variables:

- The **`PA_KUBE_STORAGE_DYNAMIC_PROVISIONING`** variable is removed.
- The **`PA_KUBE_VOLUME_TYPE`** variable is updated in the `paw.env` file.

These changes are applied automatically by the `start.sh` script during the update.

This update applies only to local deployments of Planning Analytics Workspace on OpenShift.

## What's new in modeling - 2.0.105

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.105.

### Change the default data type for new TI process parameters

A new preference setting in the Workbench process editor allows developers to choose the default data type (**String** or **Numeric**) for newly-added parameters in TurboIntegrator (TI) processes.

Previously, all new parameters were set to the **Numeric** data type by default. This meant that developers needed to take additional steps to change the data type to **String**, which is a more commonly-used data type in several development scenarios. Developers can now set their preferred default parameter type, streamlining the process creation workflow.

To configure this setting:

1. Click the **Preferences** button.

   You can also click your user name on the Planning Analytics Workspace title bar, then click **Profiles and Settings**.
2. Click **Settings**, then navigate to the **Process editor** section.
3. For the **Parameter type** field, choose either the **Numeric (default)** or **String** radio button option.

All new parameters added to TI processes will use the selected data type by default.

You must have the **Modeler** role to use this feature.

## TurboIntegrator process debugger shows variable value in hover help

The TurboIntegrator (TI) process debugger now allows modelers to view variable values by simply hovering over them during a debug session.

This feature improves the usability of the existing debugger by providing quick, in-context access to variable values without needing to reference the variable list in the debug panel. The existing debug panel remains unchanged and continues to display all variable values as before.

To use this feature, open a TI process in the process editor within the modeling workbench and run the process in debug mode. Hover your mouse over any variable to see its current value.

## Database explorer displays hierarchy element count

The Database explorer in the modeling workbench now displays the number of elements in each hierarchy of a dimension.

When viewing a dimension in the Database explorer, modelers can now see the element count for each hierarchy. This provides a quick reference for understanding the size and structure of hierarchies without needing to open them individually. Note that the element count might be less than the number of members in the hierarchy.

To view the hierarchy element count, open the Database explorer in the modeling workbench and select a dimension to view its hierarchies.

You must have the **Modeler** role to use this feature.

## Database explorer displays process data source type

The Database explorer in the modeling workbench now displays the data source type for each TurboIntegrator (TI) process, making it easier to identify and manage processes based on their input sources.

When viewing TI processes in the Database explorer, modelers can now do the following:

- See the data source type directly in the list. For example, None, Cube View, or ODBC.
- Sort and filter processes by data source type for easier navigation and organization.
- Use a new shortcut button (located between the filter and refresh icons) to quickly access the ODBC data source configuration tool. This links directly to the feature for modifying connection settings across multiple processes that use ODBC data sources.

To view the data source type, open the Database explorer in the modeling workbench (Modeler role required) and navigate to the list of TI processes.

You must have the **Modeler** role to use this feature.

### TM1 Server log provides additional details (Planning Analytics as a Service only)

The TM1 Server log viewer in the modeling workbench now includes enhanced details for each log entry, helping developers and administrators better understand server activity.

Each log entry now includes a **More** link that reveals additional context and information related to the event. This enhancement improves visibility into server operations and simplifies troubleshooting by reducing the need to cross-reference external logs and allows more log details to be reviewed and analyzed.

Access this feature from the TM1 Server log viewer:

1. Open a modeling workbench.
2. Right-click the database on the **Databases** tree.
3. Click **Logs**, then select **Server log**.
4. Click the **More** link next to any log entry.

### Drive explorer uses web browser password management (Cloud only)

The Drive explorer now supports web browser password management for credential fields.

Previously, modelers were required to manually enter their file share username and password each time they accessed local files through Drive explorer. Web browsers can now prompt users to save these credentials, streamlining access and reducing repetitive input.

You do not need to update your Drive explorer configuration to enable this feature. However, you must configure your web browser to allow credential storage.

## What's new in administration - 2.0.105

The following new and changed administration features are available in Planning Analytics Workspace 2.0.105.

### Configure the HTTPSessionTimeoutMinutes parameter in Planning Analytics Administration

You can now configure the **`HTTPSessionTimeoutMinutes`** database configuration parameter from the Planning Analytics Administration interface to manage REST API session timeouts for TM1 databases.

The **`HTTPSessionTimeoutMinutes`** parameter defines the duration of REST API sessions, in minutes, before they time out. Adjust this setting to accommodate third-party tools that require longer-lasting REST API connections to TM1 databases.

This setting is available on the **Configuration** page for your database in the Planning Analytics Administration interface:

1. From the **Databases** page, click the **Databases** tab.
2. In the list of databases, click the database for which you want to set configuration parameters.
3. Click the **Configuration** tab.
4. Navigate to the **Access** section.
5. From the drop-down list, click **HTTP**.
6. Set the new value for the **SessionTimeout** parameter.
7. Click **Apply**.

### Configure memory limit parameters in Planning Analytics Administration

You can now configure two new database configuration parameters (**`MaximumGlobalQueryMemorySizeMB`** and **`MaximumSingleQueryMemorySizeMB`**) in the Planning Analytics Administration interface to manage memory usage for MDX queries.

The following parameters are introduced to manage memory usage for MDX queries:

**MaximumGlobalQueryMemorySizeMB**
: The maximum memory, in MB, that an individual query can use before it is terminated. Use this parameter to set a global memory limit for all concurrent MDX queries. This includes memory across the MDX engine, intermediate calculations, and final results. The default value is 0 (disabled).

**MaximumSingleQueryMemorySizeMB**
: The total amount of memory, in MB, that all running queries can consume in total. Use this parameter to set a memory limit for individual MDX queries. If the total amount of consumed memory exceeds this threshold, the sysem terminates one or more queries, starting with the query that caused the overflow. The default value is 0 (disabled).

These parameters can help prevent excessive memory consumption by large or complex queries, which improves system stability and performance.

These settings are available on the **Configuration** page for your database in the Planning Analytics Administration interface:

1. On the Databases page in the Planning Analytics Administration interface, click the **Databases** tab.
2. In the latest of databases, click the database for which you want to set configuration parameters.
3. Click the **Configuration** tab.
4. Navigate to the **Performance** section.
5. From the drop-down list, click **Memory**.
6. Locate the parameter that you want to modify (**`MaximumGlobalQueryMemorySizeMB`** or **`MaximumSingleQueryMemorySizeMB`**), then set a new parameter value.
7. Click **Apply**.

**Note:** These parameters are only available after upgrading to TM1 Version 12.5.

### Increased LCM snapshot size to 100 MB

You can now create and deploy Lifecycle Management (LCM) snapshots up to 100 MB in size.

Previously, LCM snapshots were limited to 25 MB, and attempts to import or deploy larger snapshots would fail. While the existing limit of 100 assets per snapshot remains in place, the maximum file size is increased to 100 MB. This allows for more flexibility when working with larger assets, as long as the total number of assets does not exceed 100.

**Note:** It is still possible to create snapshots with fewer than 100 assets that exceed 100 MB in size. You cannot export or import snapshots of this size.

## 2.0.104 - What's new, May 20, 2025

Learn about new features and known issues in version 2.0.104 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

# What's new in books and reporting - 2.0.104

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.104.

## Customize the color of selected and highlighted cells

You can now customize the format of selected and highlighted cells by picking the color of the background and the border of the cells. You can also select a transparent background color, which allows you to view the original color of the active cells without overwriting the colors.

Previously, when you selected cells in the cube viewer, the color of the current selection is fixed and overwrites the original color of the cells, including any conditional and hierarchy formatting. This means that you cannot view the state of any cells that you selected or highlighted. In addition, the background color of the current selection or highlights might make the text difficult to read on the cells.

You can now customize the color of the background and the border of any cell that you select and any highlighted cell that is associated with the selected cells. To view the original colors of the selected and highlighted cells, select transparent as the background color for selected and highlighted cells.

To customize the format of selected and highlighted cells, follow these steps:

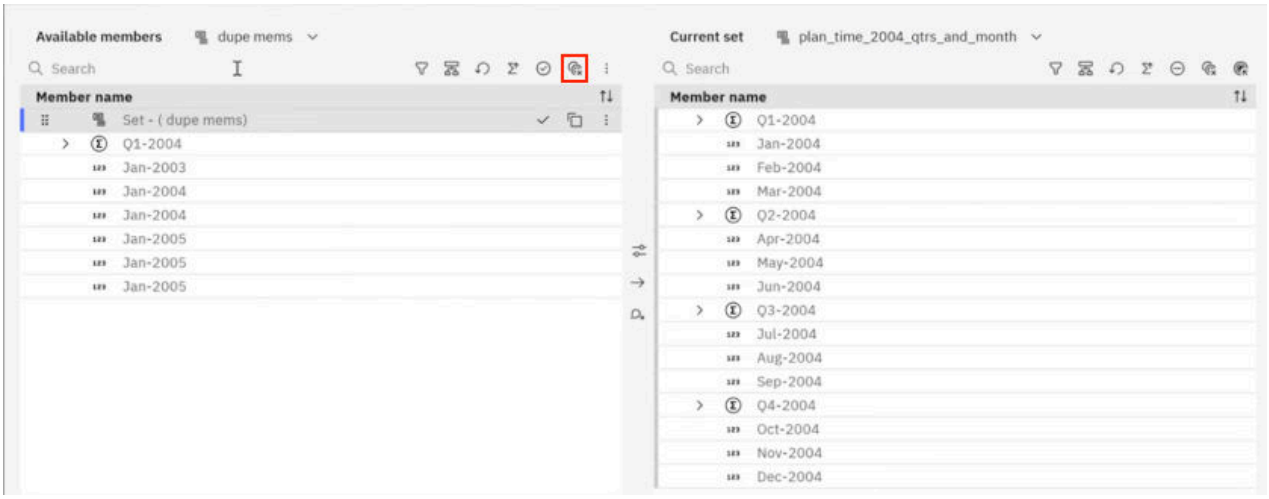
1. Select a view and click **Format manager** in the toolbar.
2. Click **Data area** on the **View** tab.
3. Under **Selection styles**, choose the colors you want for **Selected background**, **Selected border**, **Highlighted background**, and **Highlighted border**.

   To keep the original cell colors of the selected and highlighted cells, select transparent (no color) as the color for **Selected background** and **Highlighted background**.
4. Click **Apply**.

## Remove duplicate members from the Set editor

You can now remove duplicate members from the **Available members** panel in the Set editor to compare distinct members to the current set.

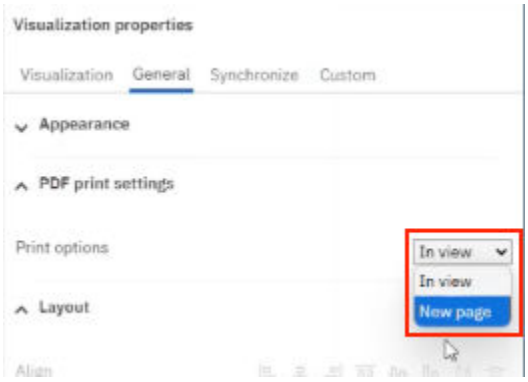
When you are in the Set editor, click the **Remove duplicated members** icon in the **Available members** panel. The **Available members** panel displays only distinct members, which you can use to compare with the current set.



## Additional options for sharing visualizations as PDF files

You can now specify additional options when generating PDF files from visualizations.

When you select **PDF print settings** for your visualizations from the **General** tab of the Visualization properties, you can now choose between **In view** and **New page**.

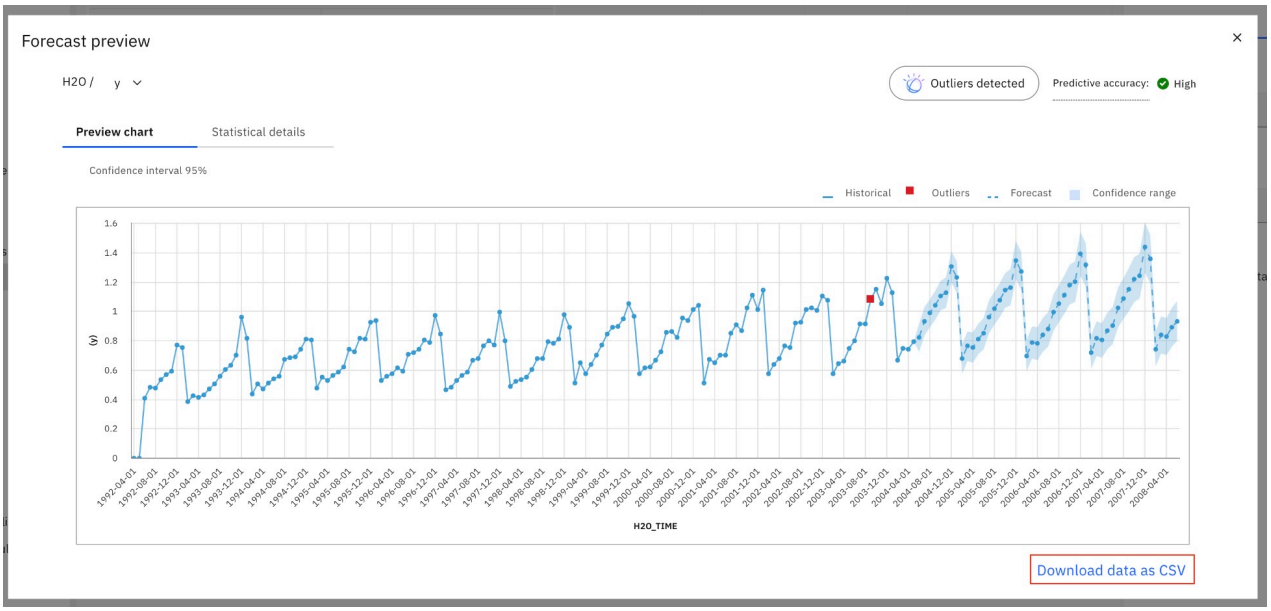



- If you select **In view**, the PDF file prints the visualization exactly as it is shown in the dashboard.
- If you select **New page**, the PDF file prints all the data from the visualization, regardless of what is shown in the dashboard. The PDF file first shows all the columns in the visualization, then all the rows. The PDF file has a new page after it prints a number of columns up to the specified number, then a new page after it prints a number of rows up to the specified number. You can also specify whether want to fit all columns in one page and all rows in one page.

## Export forecast values to a CSV file

When you are viewing a forecast preview, you can now export forecast values from the preview chart to a CSV file.

To export forecast values from the preview chart, click **Download data as CSV**.




The resulting CSV file contains data exported into the following columns:

1. Name of the tuple or the selected row.
2. Forecast periods from the time dimension.
3. Forecast slice, which consists of forecast values, high values, and low values.
4. Actual values for each of the slices and the corresponding forecast time period.

# What's new in modeling - 2.0.104

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.104.

## Specify multivariate options for baseline forecasts

You can now create multivariate types of baseline forecasts. Multivariate forecasts allow you to specify a maximum of two variables for the baseline forecasts.

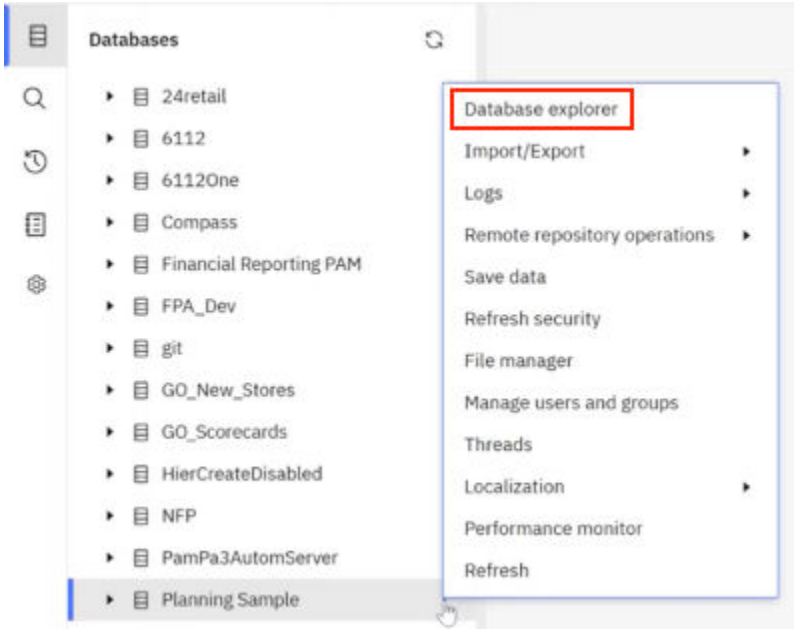
When you create a baseline forecast, define the forecast as a multivariate forecast in the **Details** screen by clicking the **Univariate** or **Multivariate** icon until it is green with the **Multivariate** label. 


If you specify a multivariate baseline forecast, you are prompted to add the multiple variables in the **Multivariate** page.

## Use the new Database explorer to view and perform actions on database objects

You can now view information and perform common bulk actions on database objects with the new **Database explorer**.

To open the Database explorer, right-click the database on the modeling workbench **Databases** tree, then click **Database explorer**.




The Database explorer opens a **Properties** tab with the name of the database, which displays a table with the main TM1 objects that you can view, along with the numbers of each object in the specified database. Click the item count to open a table that displays all of the specified objects in the database. You can select multiple objects in the table and perform a bulk action on all selected objects. For example, you can choose to unlock multiple cubes.

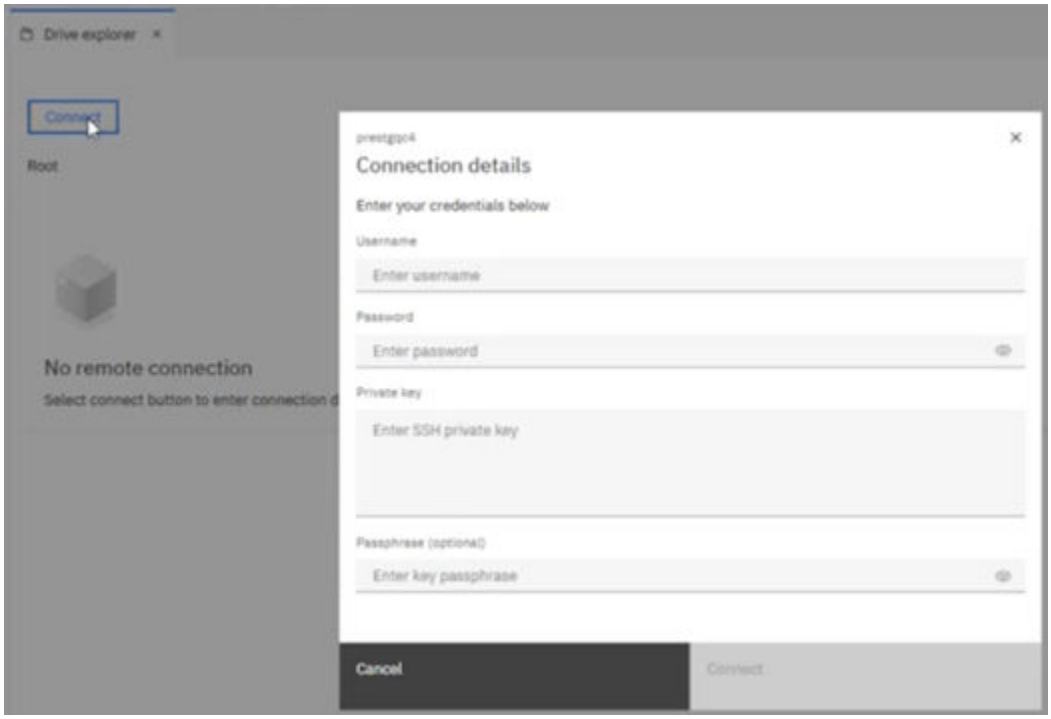
# What's new in administration - 2.0.104

The following new and changed administration features are available in Planning Analytics Workspace 2.0.104.

### Control user access to shared folders in Drive explorer

Drive explorer now uses the Access Control List (ACL) to control user access to shared folders in the S: / drive.

The existing ACL that is used to control user access to the shared folder is now used to also control user access to files in Drive explorer. If you click **Connect** on the Drive explorer, you are prompted for user credentials in the **Connection details** dialog prompt.




The ACL determines which shared files and folders that your user account can access in Drive explorer. Previously, Drive explorer did not control user access to shared folders.

### Show API key expiration date (Cloud only)

The API key table now shows the expiration date of API keys in an additional column.

If you generate API keys using MCSP, these keys are valid for 12 months. After you create an API key, the expiration date of the key now appears in the existing API key table in a new column.

## Known issues - 2.0.104

The following are known issues in Planning Analytics Workspace 2.0.104. These issues will be resolved in a future release.

### Users with Delete permission cannot delete files in Drive explorer

Drive explorer uses the Access Control List (ACL) to control user access to shared folders in the S: / drive. With ACL, users with the Delete (d) permission cannot delete files in the Drive explorer.

Any user with Write (w) permission will be able to delete files or folders, even though the delete permission is set to deny.

# 2.0.103 - What's new, April 22, 2025

Learn about new features and known issues in version 2.0.103 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.103

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.103.

### Create views by using natural language

You can now create views in books with the new **Data explorer** feature using natural language. The **Data explorer** feature uses generative AI to analyze your query and matches it to the relevant data in TM1 to recommend views. An explanation for the logic behind the recommendation accompanies each recommended view. You can make changes to the view and add it to the book or insert it as-is.

The initial release of **Data explorer** is available in English only.

**Important:** The **Data explorer** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM® sales representative or go directly to your IBM account.

Creating views in the **Data explorer** is a two-part process:

**Analyze cube**
For **Data explorer** to recommend views, you need analyzed cubes. Cube analysis takes place in the modeling workbench, which allows AI to review the structure of a cube and understand the data.

**Submit query**
Once cube analysis is complete, you can go to a book and enter your query in the **Data explorer** to get recommended views.

### Creating views in Data explorer

Follow these steps to run a cube analysis and then create a view in **Data explorer**:

1. In the modeling workbench data tree, go to the cube that you want to analyze.
2. From the cube's menu, click **AI operations > Analyze cube** and click **Analyze**.

   **Note:**

   - You need to have access to CubeAttributes, DimensionAttributes, and the corresponding SubsetAttributes control cubes.
   - You can check the status of a cube analysis by clicking **AI operations > Analyze log**.
3. After the cube analysis completes, go to the book where you want to create the view.
4. Click **AI operations** and select **Data explorer**.
5. Select the database and analyzed cube with the data that you want to use to create a view.

**Note:** If you choose **Auto-select** for the analyzed cube, the most relevant cube is automatically selected to create a view, based on your query.

6. Enter your query in natural language. For example, you can enter `Show me sales performance trends over the last year` or `Show me 2024 sales of mobile devices in north america`.
7. Click **Submit**.

A view generates along with an explanation for the recommended view and the cube that is chosen to generate the view. You can add the view to the book in edit mode.

You can also make certain changes to the view such as switch rows and columns, move dimensions to the context area, edit the set, and change levels.

If there are other analyzed cubes with data that matches your query, they are listed next to **Alternative cubes**. Click an alternative cube to see a recommended view from that cube.

For more information, see Create views with Data explorer using natural language.

## Use the Format manager to apply formatting to hierarchies

You can now create and apply formatting rules to hierarchies in views. Use the new **Hierarchy** tab in the **Format manager** to create a formatting rule and apply it to a hierarchy.

To create a hierarchy formatting rule, right-click a member in the view and select **Format manager > Hierarchy** or select a view and in the **Format manager**, go to the **Hierarchy** tab. Then, follow these steps:

1. Select a dimension and hierarchy and click **Add format**.
2. Define the rule by selecting the condition type:
   a. **Base format** - Applies the rule to all of the members in the hierarchy.
   b. **Level** - Applies the rule to members of a selected hierarchy level.
   c. **Member attribute** - Applies the rule to members with a specific attribute value.
   d. **Member name** - Applies the rule to members that contain a specific value in their name.
3. (Optional) Specify formats for the data cells and header cells.
4. Click **Apply** to save the format.

If you have Member and View formats defined along with Hierarchy formats, the formatting rules apply in the following order:

1. Member formats
2. Hierarchy formats on the Columns
3. Hierarchy formats on the Rows
4. View formats

If you have multiple hierarchy formatting rules, the first rule has the highest priority. You can rearrange the priority of a formatting rule by dragging it up or down the list of rules.

For more information, see Format cube views with the Format manager.

## Set cell background and font colors by using member and hierarchy formats

You can now set the cell background and font color for data cells and header cells with member and hierarchy formats.

You can define a member format to have a specific background and font color and also use hierarchy formatting to define the cell and font colors based on the leaf level.

The following example uses hierarchy formatting to set the background color of the data and row headers based on their level. The leaf-level data values are displayed in gray font color.

| region | 1 Quarter | Jan | Feb | Mar | 2 Quarter | Apr | May | Jun |
|---|---|---|---|---|---|---|---|---|
| World | 21519.00 | 6713.00 | 7593.00 | 7213.00 | 16802.00 | 6409.00 | 5258.00 | 5135.00 |
| Europe | 17954.00 | 5759.00 | 6275.00 | 5920.00 | 13808.00 | 5286.00 | 4347.00 | 4175.00 |
| Scandinavia | 898.00 | 273.00 | 316.00 | 309.00 | 679.00 | 263.00 | 212.00 | 204.00 |
| Denmark | 248.00 | 76.00 | 87.00 | 85.00 | 189.00 | 74.00 | 59.00 | 56.00 |
| Norway | 293.00 | 87.00 | 104.00 | 102.00 | 230.00 | 88.00 | 72.00 | 70.00 |
| Sweden | 357.00 | 110.00 | 125.00 | 122.00 | 260.00 | 101.00 | 81.00 | 78.00 |
| Benelux | 719.00 | 336.00 | 382.00 | 1.00 | 819.00 | 308.00 | 262.00 | 249.00 |
| Belgium | 325.44 | 149.00 | 176.00 | 0.44 | 371.00 | 139.00 | 115.00 | 117.00 |
| Luxemburg | 33.05 | 16.00 | 17.00 | 0.05 | 37.00 | 14.00 | 12.00 | 11.00 |
| Netherlands | 360.51 | 171.00 | 189.00 | 0.51 | 411.00 | 155.00 | 135.00 | 121.00 |
| Islands | 2692.00 | 870.00 | 935.00 | 887.00 | 2034.00 | 773.00 | 646.00 | 615.00 |
| Great Britain | 2450.00 | 794.00 | 851.00 | 805.00 | 1849.00 | 701.00 | 588.00 | 560.00 |
| Ireland | 242.00 | 76.00 | 84.00 | 82.00 | 185.00 | 72.00 | 58.00 | 55.00 |
| Central Europe | 9446.00 | 2960.00 | 3166.00 | 3320.00 | 7143.00 | 2727.00 | 2231.00 | 2185.00 |
| France | 4503.00 | 1410.00 | 1526.00 | 1567.00 | 3334.00 | 1276.00 | 1025.00 | 1033.00 |
| Germany | 4943.00 | 1550.00 | 1640.00 | 1753.00 | 3809.00 | 1451.00 | 1206.00 | 1152.00 |
| Iberia | 1206.00 | 381.00 | 416.00 | 409.00 | 884.00 | 336.00 | 277.00 | 271.00 |
| Portugal | 201.00 | 63.00 | 70.00 | 68.00 | 147.00 | 55.00 | 48.00 | 44.00 |
| Spain | 1005.00 | 318.00 | 346.00 | 341.00 | 737.00 | 281.00 | 229.00 | 227.00 |

To apply member or hierarchy formats, right-click a member and select the type of format you want to apply. Or, use the **Format manager** in the toolbar to access both member and hierarchy formatting options.

For more information, see Format cube views with the Format manager.

## Enhancements to the parameter control types for TurboIntegrator processes

Enhancements to the parameter control types for TurboIntegrator processes were implemented in Planning Analytics Workspace 2.0.103 . The **Dimension list** control type was renamed to **Member list** and you can now choose if a member name or member unique name is passed to the process parameter. A new control type, **Set list**, now lets you pass a named subset or a selection in a TurboIntegrator process.

**Member list**

The **Member list** presents a list of dimension set members from which the user can choose the parameter value.

To define the **Member list** parameters, click the **Edit** icon. Pick the dimension that contains the set you want to appear in the list and the hierarchy that contains the set you want to use.

You can also choose a subset that contains the members that you want to appear in the list. If you don't select a subset, then it is the member name that passes and the user clicking the Action Button will select from all of the members without aliases.

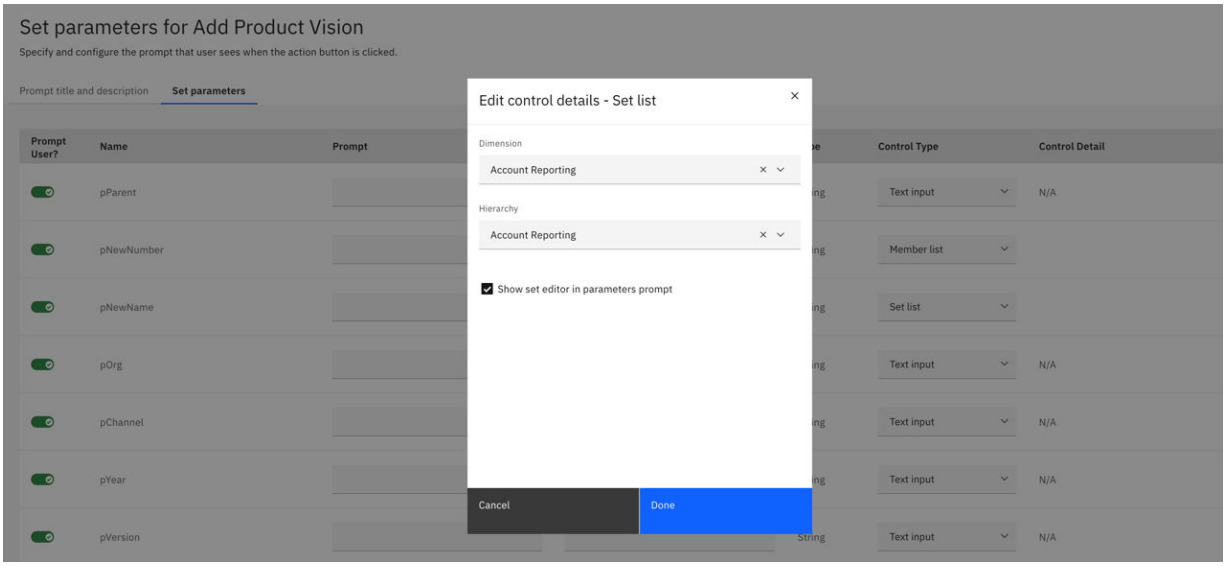
You can pass either the member name or member unique name in the TurboIntegrator process. The member name passes by default.

**Set list**

The **Set list** passes a subset to the TurboIntegrator process.

To define the **Set list**, click the **Edit** icon and pick the dimension and hierarchy. The user can then choose from the list of subsets in the hierarchy when they click the Action Button to run the process.

You can choose to show the set editor in the parameters prompt which allows the user to open the set editor and generate a selection instead of using one of the named subsets.




# What's new in modeling - 2.0.103

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.103.

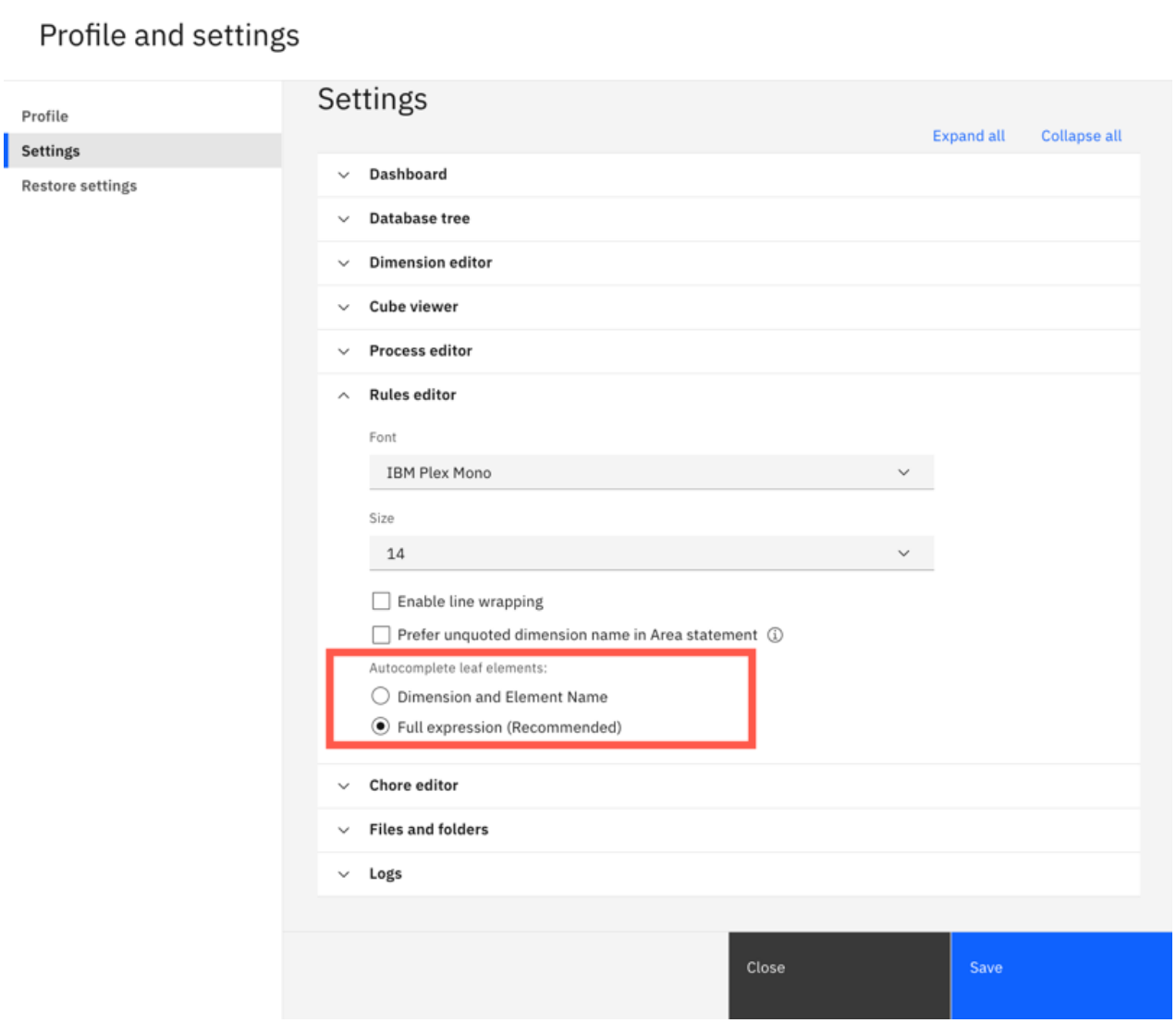
## New auto-complete options in the rules editor in workbench preferences

New options are available in the modeling workbench preferences that specify how leaves display in auto-complete when editing rules. You can choose if you want to see just the dimension name and element in auto-complete for a leaf or the full expression, which includes the hierarchy. Auto-complete displays the full expression by default.

Elements now also appear in a different color which makes it easier to differentiate them from the dimension name.

To set auto-complete preferences for the rules editor, follow these steps:

1. Click the **Preferences** icon in the modeling workbench.
2. Select an option for **Autocomplete leaf elements** under **Settings > Rules editor** and click **Save**.




## Change the member type to consolidated in the dimension editor

You can now change the properties of a leaf member from a numeric or string type to consolidation without setting children in the modeling workbench. Previously, you could only switch the leaf member type from a numeric to a string and vice versa.

**Important:** The feature requires TM1® 11.8.29 or a later version, or TM1® 12.4.8 or a later version.

To change the member type in the modeling workbench:

1. On the **Databases** tree, right-click the dimension you want to work on and click **Edit dimension**.
2. Click the **Members** tab on the dimension editor.
3. Select the desired member type in the **Type** column of the leaf member that you want to change.

## Create shortcuts to all TM1 objects in the modelling workbench

The shortcut feature in the Planning Analytics Workspace modelling workbench now supports all object types in the TM1 database, allowing you to easily group, organize, and access modelling objects. Shortcuts to TM1 objects now save in the **Applications** folder in the data tree. This folder was previously called **Websheets** and only supported shortcuts or links to websheets and native views.

**Note:** This feature requires TM1 versions 11.8.29 or later, or TM1 v12.

You can create shortcuts for the following objects:

- Dimensions
- Cubes
- MDX views
- Native views
- Processes
- Chores
- Websheets
- Subsets

  **Note:** In TM1 v11, you cannot create shortcuts for subsets with hierarchy names that are different from the dimensions.

To create a shortcut to an object from the database tree, right-click the object and select **Add shortcut**. Select a destination folder or create a new folder within the **Applications** folder. If the object is already open in the workbench, you can create a shortcut from the workbench tab for each supported object type with the **Add shortcut** icon.

After a shortcut is created, you can rename it in the database tree. You can also delete a shortcut.

**Note:** Deleting a shortcut doesn't delete the TM1 object. Similarly, if you delete an object from the database tree, the related shortcut isn't deleted. When you use a shortcut for a deleted TM1 object, you'll get an error notifying you that the object could not be found.

You can apply security to the shortcuts and set them to be public or private.

To use an object through the shortcut, simply double-click or drag the object to the workbench.

### Accessing TM1 object shortcuts in books

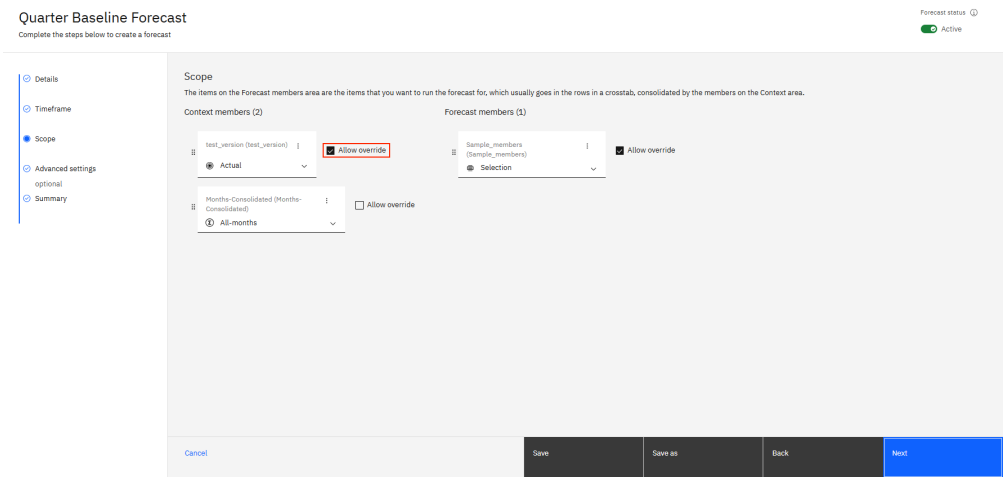
You can access all shortcuts, except those that are for processes and chores, in Planning Analytics Workspace books, from the **Applications** folder in the database tree.

## Increase the flexibility of baseline forecasts by allowing users to override default context members

An administrator or modeler can now set an option that enables users to override default context members and specify a custom scope when running a baseline forecast.

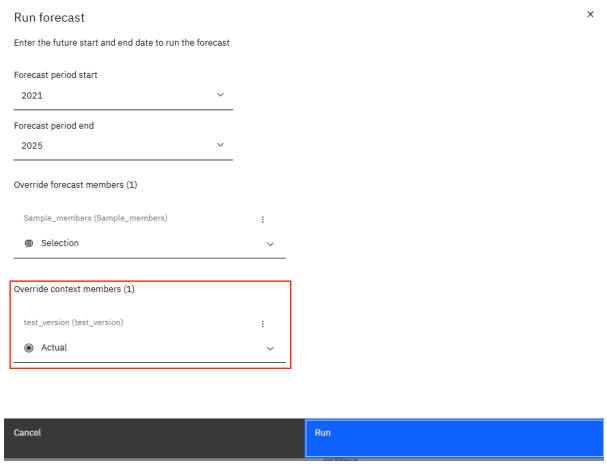
This makes baseline forecasts more flexible for users and simplifies maintenance for baseline forecast creators, who no longer need to create and maintain multiple forecasts with similar but varying scopes.

To enable users to override default context members and specify a custom scope, enable the **Allow override** field for when you define the scope of a baseline forecast.



Previously, you could override the default only for forecast members.

When a user runs a baseline forecast with the **Allow override** option enabled, they can use the **Override context members** option to pick an existing member or level for the forecast.



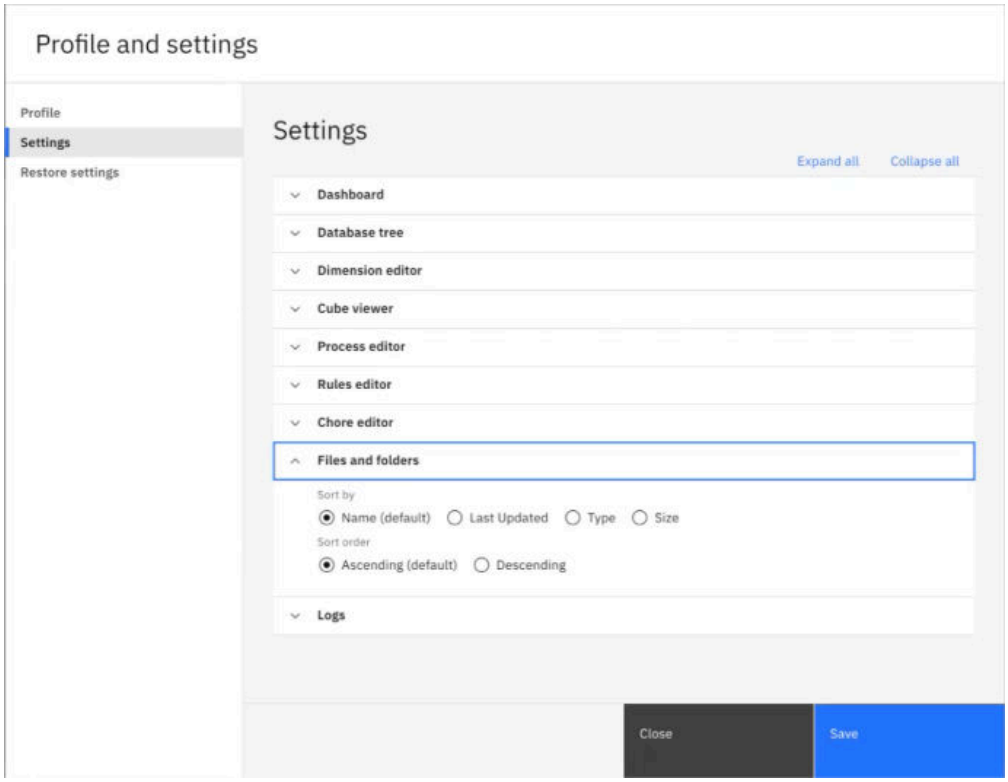

Though the user specifies a set or level that differs from the default defined in the baseline forecast, the forecast definition remains unchanged. The custom selection is applied only at run time.

## Sorting options for items in File manager and Drive explorer

You can now specify your sort order preferences for items in the File manager and Drive explorer.

To specify the sort order of items in the File manager and Drive explorer, navigate to **Files and folders** in your profile settings.

1. In the **Personal menu**, click **Profile and settings**.
2. Click **Settings** and under **Files and folders**, select the sort column and the sort order. By default, File manager and Drive explorer sort by the **Name** column in ascending order.




3. Click **Save**.

The File manager and Drive explorer now show items sorted in the order that you specified.

# What's new in administration - 2.0.103

The following new and changed administration features are available in Planning Analytics Workspace 2.0.103.

## Set database thresholds and alerts in TM1 Database 12

You can now set database thresholds and alerts in TM1 Database 12 for Memory usage and CPU usage metrics.

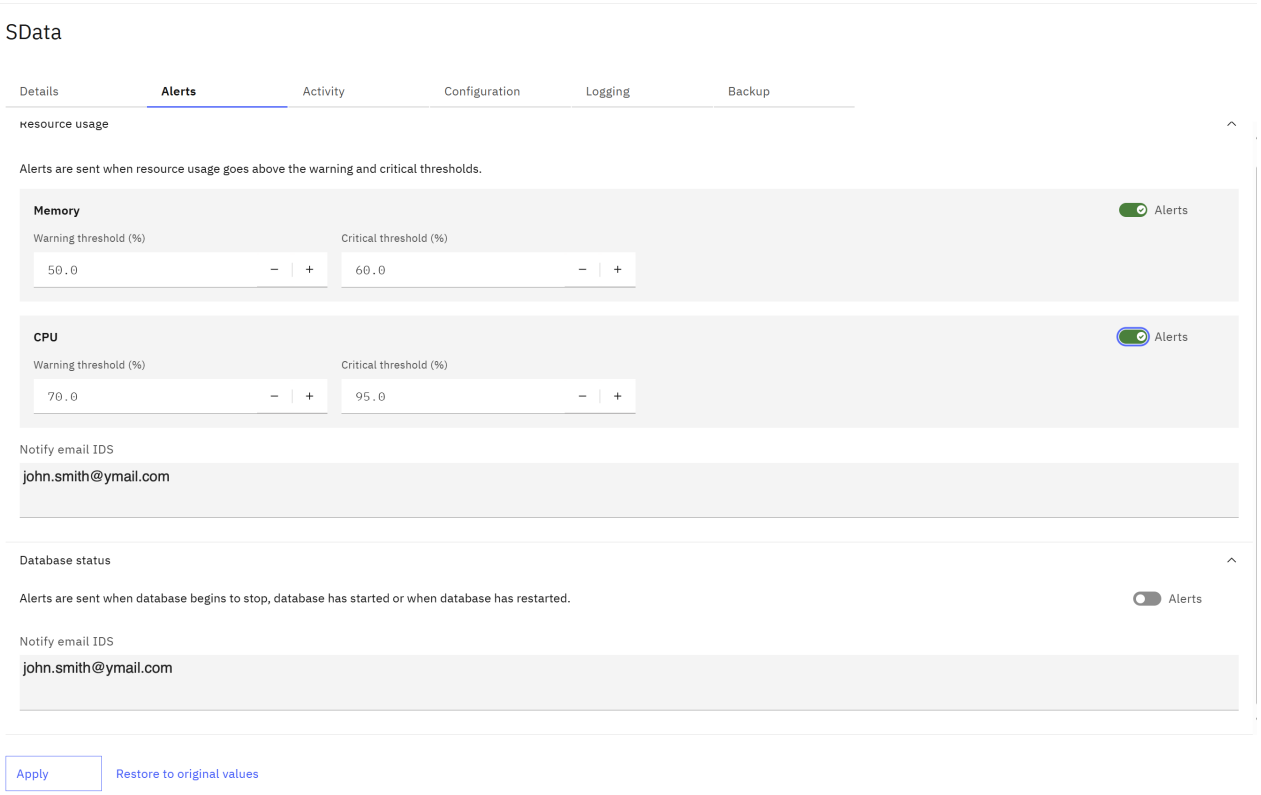
When you set threshold limits, you define the parameters for the concern and critical states of the following metrics for an individual database:

- Memory usage - Measures database memory usage in GBs.
- CPU usage - Measures database CPU usage as a percentage of total CPU capacity on the system.

You can define the **Warning threshold** and **Critical threshold** for both of these metrics.

When the database metric value falls below the Warning threshold, the database is in a healthy state. When the metric is at or above the Warning threshold, but below the Critical threshold, the database is in a concern state. When the metric is at or above the Critical threshold, the database is in a critical state.

You can choose to be alerted by email when a database metric reaches a critical state or when the database starts, stops, or restarts.



### Import users to Planning Analytics Workspace from the TM1 authentication database (Local only)

Administrators can now quickly import users from the TM1 database that is used for authentication into Planning Analytics Workspace.

You need to have a TM1 authentication database to use this feature. The database that is used for authentication is identified by the icon. The icon is displayed only on Planning Analytics Administration Local and only when Planning Analytics Workspace is configured to authenticate against a specific Planning Analytics database.

**Important:** For the authentication database to be correctly identified in Planning Analytics Administration, the TM1LoginServerURL configuration parameter for Planning Analytics Workspace Local must be specified with an IP address and port. The IP address must match the address that is specified by the IPAddressV4 parameter in your `tm1s.cfg` file. The port number must match the port that is specified by the HttpPortNumber parameter in your `tm1s.cfg` file.

```
TM1LoginServerURL=https://
IPAddressV4_from_tm1s.cfg:HttpPortNumber_from_tm1s.cfg
```

If you have multiple tenants (environments), each tenant must be configured so that the authentication database is available from each tenant.

To import users from the TM1 authentication database, follow these steps:

1. In Planning Analytics Administration, go to **Users and Groups**.
2. On the **Users** tab, select the role and user that you want to import from the TM1 authentication database.
3. Click **Import**.

**Note:** You can import only users, not user groups.

## What's new in Planning Analytics Workspace Local - 2.0.1032.1.10

The following new and changed administration features are available in Planning Analytics Workspace Local 2.0.103 .

### Run Planning Analytics Workspace Local with Podman rootless containers

You can now run Planning Analytics Workpace Local by using Podman rootless containers. Previously, Podman rootless containers were not supported, although the user installing and running Planning Analytics Workspace could be non-root. Now, if a user is configured to access both the root and rootless (user-mode) Podman socket, the rootless socket takes priority over the root socket.

The Podman container engine supports running containers without root privileges by default. Unlike Docker, Podman does not require a central daemon, which allows containerized processes to run without elevating a user's privileges. This adds a layer of security because the container engine, orchestrator, and the runtime can all execute in the user's context and are therefore limited to the user's privileges.

Without a rootless setup, containerized processes might still run as limited users, but the container engine holds root privileges and containers can be granted elevated access.

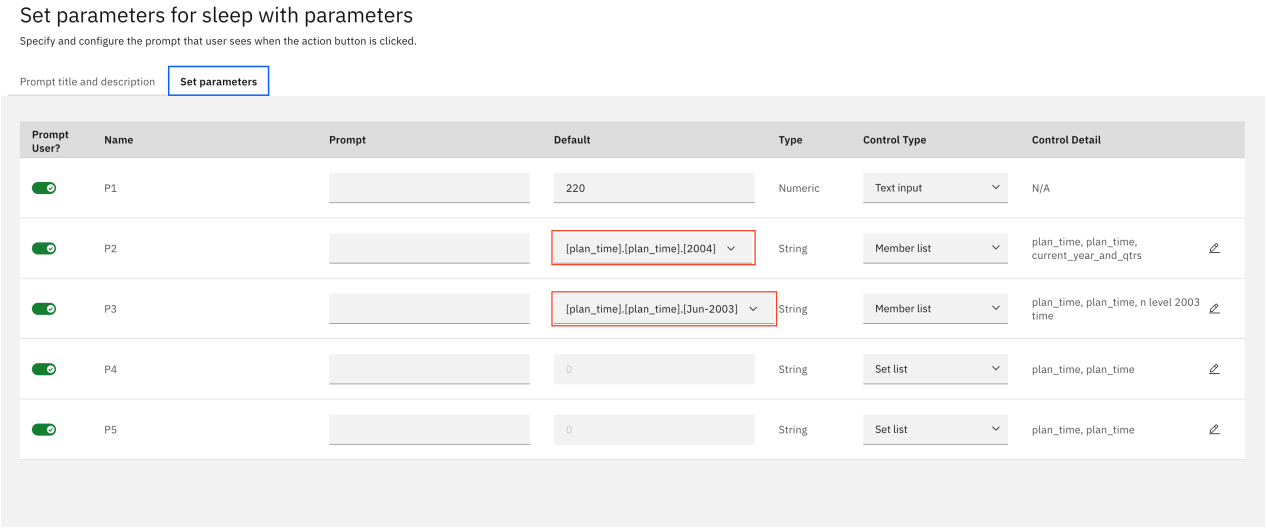
For information about enabling rootless Podman, see the Prerequisites for installing Planning Analytics Workspace Local.

## Known issues - 2.0.103

The following are known issues in Planning Analytics Workspace 2.0.103. These issues will be resolved in a future release.

### Known issue - Member unique name is displayed incorrectly

For the **Member** field, even if you select **Pass name** to pass the member name when users click the Action button, the Action button configuration page (**Set parameters**) still displays the member unique name (MUN) instead of the member name or alias.




### Workaround

To work around this issue, for each action button that you saved where you specified **Pass member unique name** for the **Member** field, edit the control details for member list, explicitly set **Pass member unique name** for the **Member** field, then save the book to preserve this setting.

# 2.0.102 - What's new, March 19, 2025

Learn about new features and known issues in version 2.0.102 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.102

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.102.

### AI-generated exploration analysis (Cloud only)

A new **Exploration analysis report** feature uses AI to generate an analysis of a selected view in the form of a PDF report. The generated report includes an executive summary followed by an impact analysis and outlier analysis for the exploration, allowing you to take faster business decisions.

The initial release of **Exploration analysis report** feature is available only in Planning Analytics Workspace Cloud and in English only.

**Important:** The **Exploration analysis report** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM® sales representative or go directly to your IBM account.

To start an exploration analysis, follow these steps:

1. Select an exploration in a book.
2. In the toolbar, click **AI operations** and select **Exploration analysis report**.
3. After the report is generated, a notification displays from which you can open the PDF file.

The exploration analysis is run on the first 120 rows and columns of the exploration.

When you run an **Exploration analysis report**, an impact analysis and outlier analysis is performed on the exploration.

The impact analysis determines the impact of data fields on each other as well as identifies and quantifies relationships across the data. Generative AI interprets and summarizes the analysis to describe the direct and indirect impact of the data fields on each other, as well as any dependencies and relationships between them.

An outlier analysis on the exploration identifies data points that deviate significantly from the norm, potentially indicating errors or anomalies. AI-generated insights are available for any outliers that were detected, helping you to investigate anomalies and improve the quality of your data.

### AI-generated summary

The **Exploration analysis report** feature uses granite-3-8b-instruct from the Granite family of IBM foundation models to generate the executive summary and insights from the impact and outlier analysis.

The Granite models are decoder-only models that can efficiently predict and generate language. These models were built with trusted data that has the following characteristics:

- Sourced from quality data sets in domains such as finance (SEC Filings), law (Free Law), technology (Stack Exchange), science (arXiv, DeepMind Mathematics), literature (Project Gutenberg (PG-19)), and more.
- Compliant with rigorous IBM data clearance and governance standards.
- Scrubbed of hate, abuse, and profanity, data duplication, and blocklisted URLs, among other things.

**Note:** IBM is committed to building AI that is open, trusted, targeted, and empowering. For more information about contractual protections that are related to IBM indemnification, see the IBM Client Relationship Agreement and IBM watsonx.ai service description.

### Tips on working with AI

While working with **Exploration analysis report**, here are few things that you can do that might lead the AI model to generate a better analysis:

- Ensure that the values in the exploration data are numeric.
- Remove spacers from rows or columns.
- Remove duplicate values.

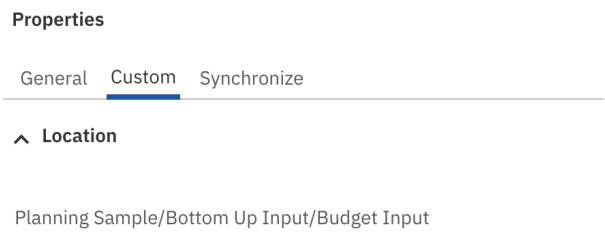
## Websheet enhancements

Learn about the enhancements that have been made to Websheets in IBM Planning Analytics Workspace.

**Websheet location**

You can now see the location of a websheet.

To view the websheet path, click **Properties** > **Custom** > **Location**.



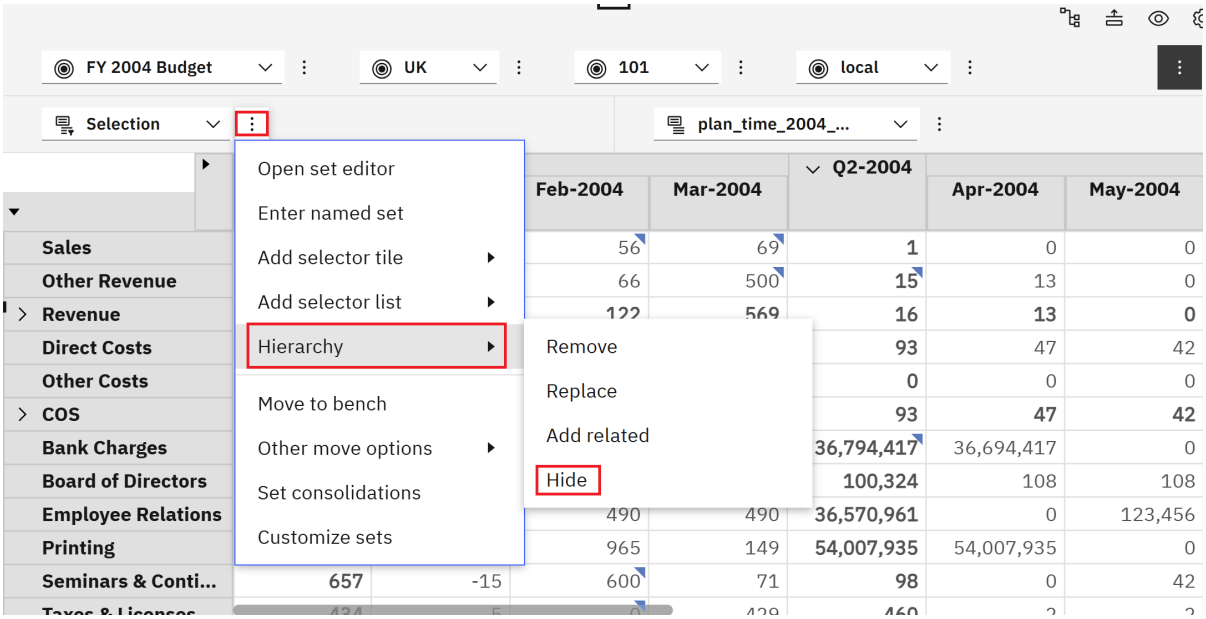

**Double-click**
You can now double-click on a websheet in the database tree to add it to the dashboard.

**Drag and drop**
You can now drag and drop a websheet onto an existing websheet to replace it.

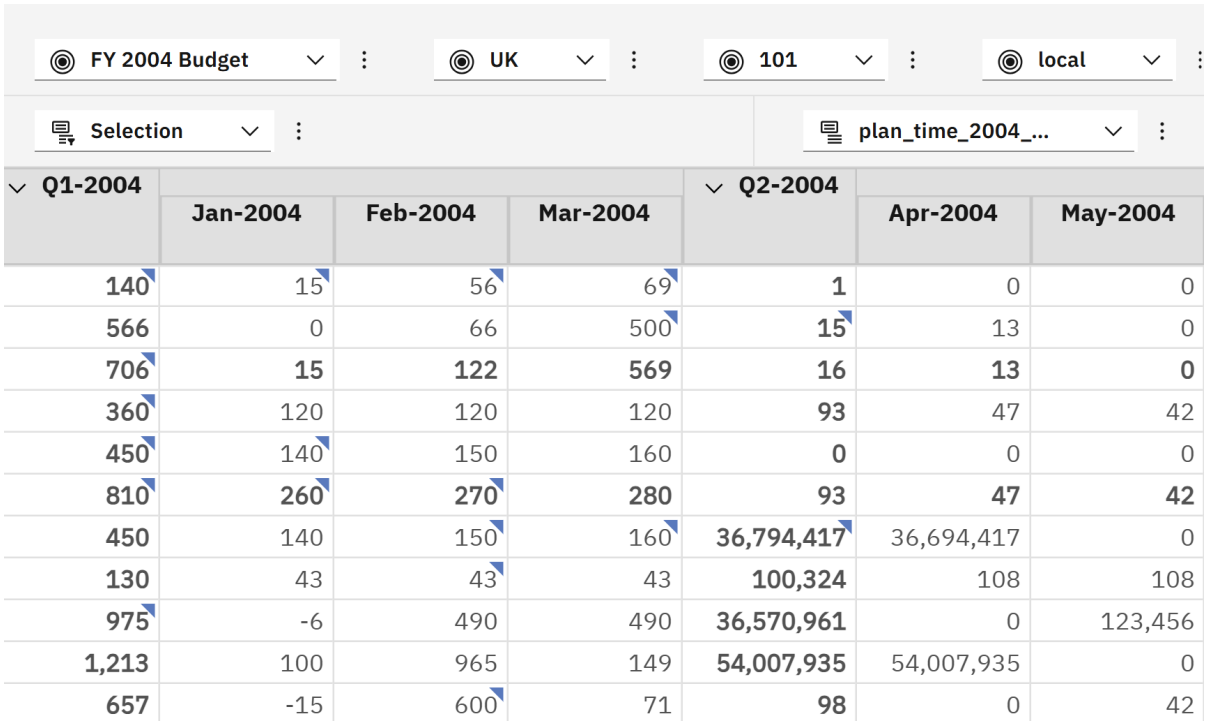
## Hide row and column headers in a view

You can hide row and column headers in a view without triggering a data update.

To hide a row or header, click **Hierarchy** > **Hide** from the dimension tile context menu.



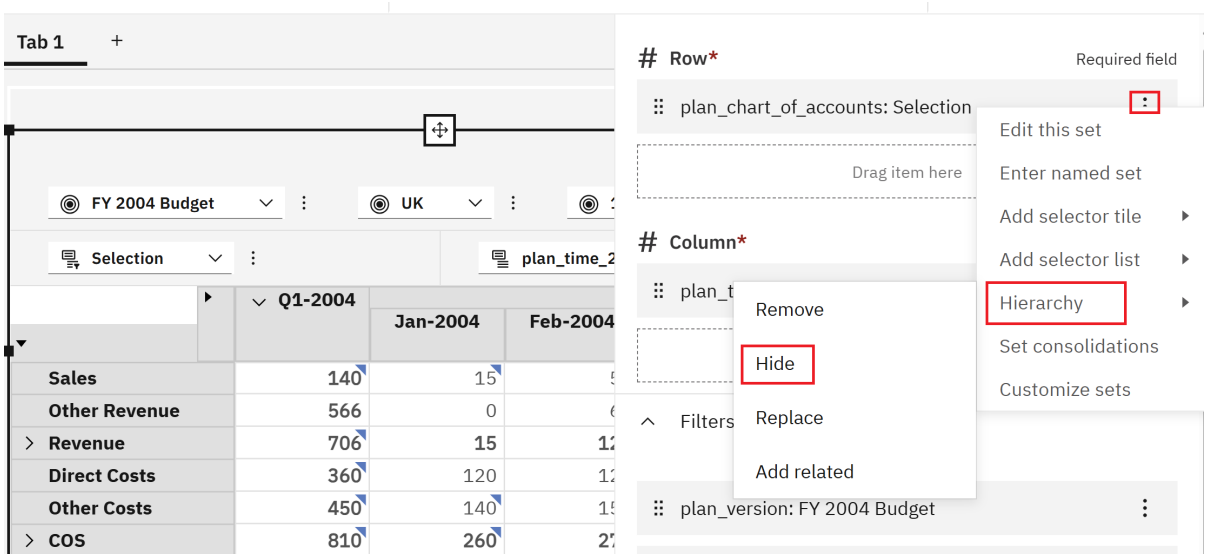

In the following image, the row headers are hidden.



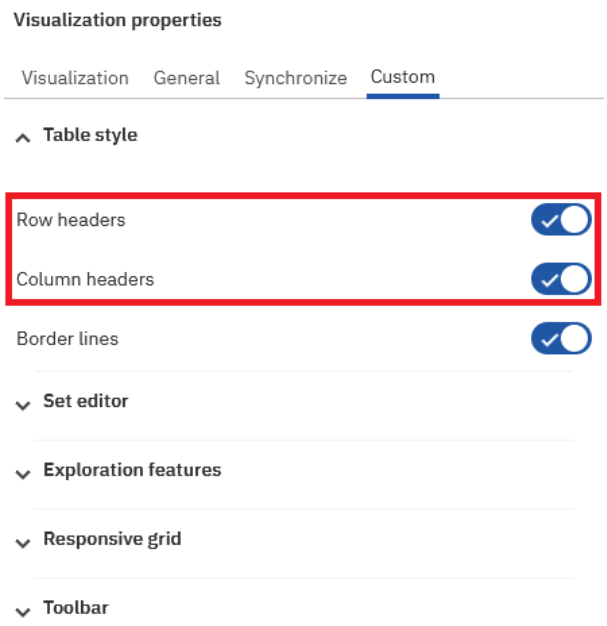

| Q1-2004 | Jan-2004 | Feb-2004 | Mar-2004 | Q2-2004 | Apr-2004 | May-2004 |
|---|---|---|---|---|---|---|
| 140 | 15 | 56 | 69 | 1 | 0 | 0 |
| 566 | 0 | 66 | 500 | 15 | 13 | 0 |
| 706 | 15 | 122 | 569 | 16 | 13 | 0 |
| 360 | 120 | 120 | 120 | 93 | 47 | 42 |
| 450 | 140 | 150 | 160 | 0 | 0 | 0 |
| 810 | 260 | 270 | 280 | 93 | 47 | 42 |
| 450 | 140 | 150 | 160 | 36,794,417 | 36,694,417 | 0 |
| 130 | 43 | 43 | 43 | 100,324 | 108 | 108 |
| 975 | -6 | 490 | 490 | 36,570,961 | 0 | 123,456 |
| 1,213 | 100 | 965 | 149 | 54,007,935 | 54,007,935 | 0 |
| 657 | -15 | 600 | 71 | 98 | 0 | 42 |

To show the row header, click **Hierarchy** > **Show** from the dimension tile context menu.

The same options are also available from the **Fields** panel.



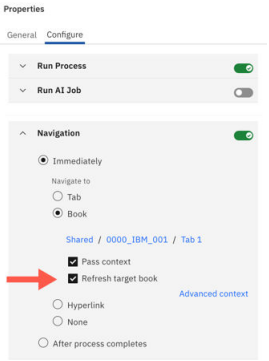

The **Hide** and **Show** context menu options are available only when the **Row headers** and **Column headers** visualization properties are turned on the **Custom** tab, as shown in the following image.




For more information, see Hide row and column headers in a view.

## Refresh the target book on navigation using an action button

A new **Refresh target book** option is available in the Action button configuration options for navigation. This option allows you to refresh a target book you are navigating to when you click an Action button, if the book is already open. Previously, if the target book was already open, you had to manually refresh the book on navigation.




When the **Pass context** option is also enabled, context passes for any items that have synchronization enabled and the target book refreshes. If **Pass context** is enabled but no context is being passed on navigation, the target book still refreshes, if it is already open.

## Zebra strips property moved to Format manager

The **Zebra strips** property for explorations is now available in the **Data area** of the **Format manager**. Previously, this property was located on the **Custom** tab under **Properties**.

The **Zebra strips** property controls row shading. Enabling it applies a shaded background to alternating rows in an exploration.

To apply **Zebra strips** to an exploration:

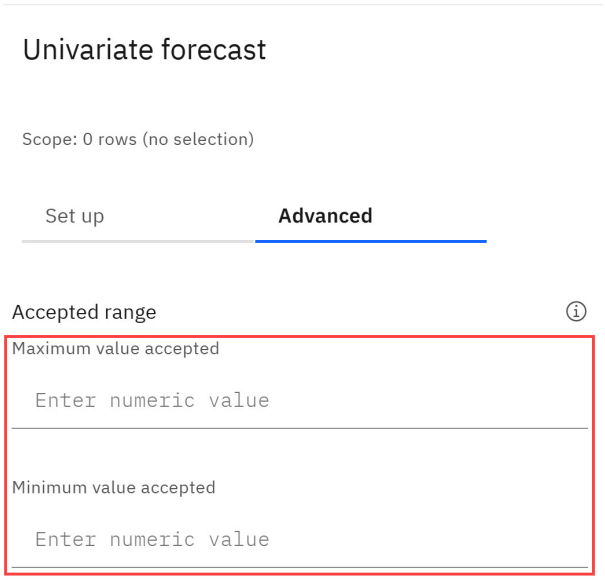
1. Select an exploration while in **Edit** mode.
2. Open the **Format manager** from the toolbar.
3. On the **View** tab, click **Data area**.

4. Under **Table styles**, select **Zebra strips**.

### Set a maximum and minimum for forecast values

You can now set a maximum and a minimum for values that are saved by a univariate or multivariate ad-hoc forecast.

There might be circumstances where you want to limit the maximum and minimum values for a forecast. For example, your business likely cannot produce a number of units in excess of manufacturing capacity or sell a negative number of units, though a forecast might predict such values. To address these circumstances, the **Advanced** tab of an ad-hoc forecast now includes an **Accepted range** section, where you can set the **Maximum value accepted** and **Minimum value accepted**.




The **Maximum value accepted** specifies the highest value that can be saved for the forecast. For example, if you specify 100 as the maximum, any forecast values that exceed 100 will be saved as 100.

The **Minumum value accepted** specifies the lowest value that can be saved for the forecast. For example, if you specify 0 as the minimum, any forecast values that are lower than 0 will be saved as 0.

**Important:** These settings apply only to the data that is saved for a forecast. The forecast preview will still show the actual generated forecast values.

Note that you can set a **Maximum value accepted** and **Minimum value accepted** for ad-hoc forecasts only; you can't specify these values for a baseline forecast.

## What's new in plans and applications - 2.0.102

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.102.

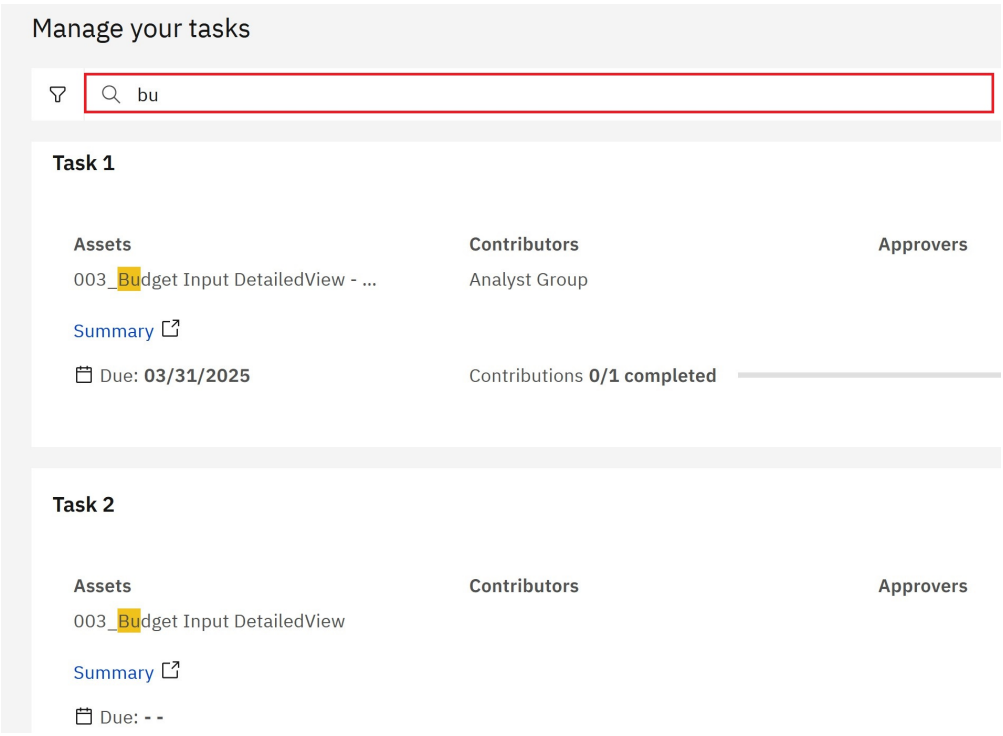
### Search and filter in plans and applications

You can search and filter on plan tasks and application sections in edit mode.

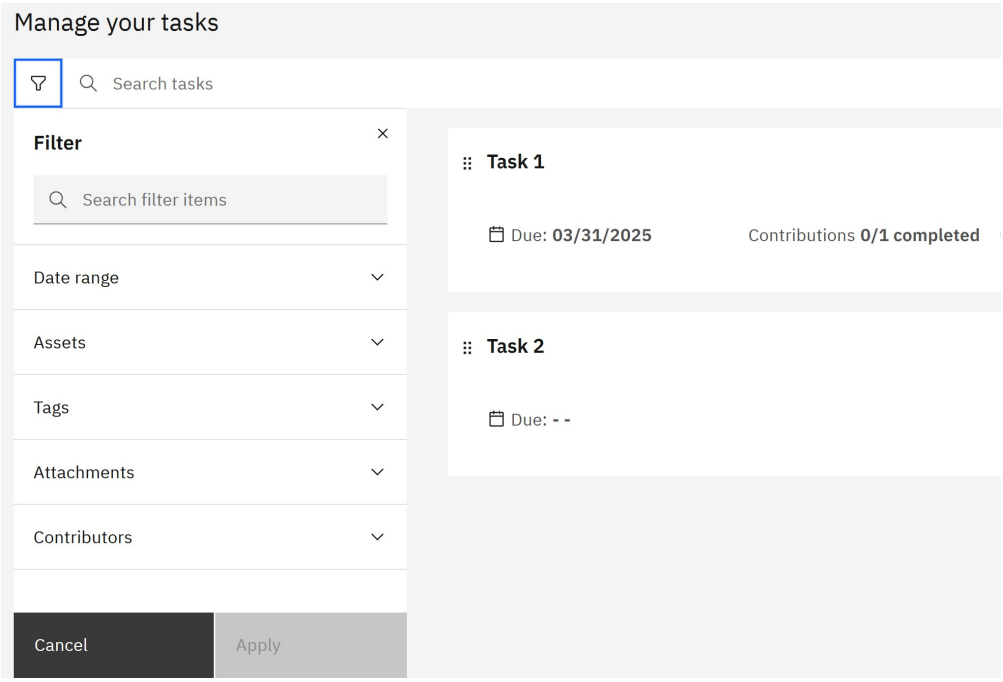
The search input field and filter button are shown before the task or section tiles.

Tasks can be searched and filtered by assets, tags, attachments, contributors, approvers, task dependencies, status, and task progress. Sections can be searched and filtered by assets, tags, attachments, and contributors.

When you type the search keyword in the search field, the task or section tiles that contain the searched items are expanded, and the search keywords are highlighted.




To filter on tasks or sections, click the filter button to open the **Filter** panel.



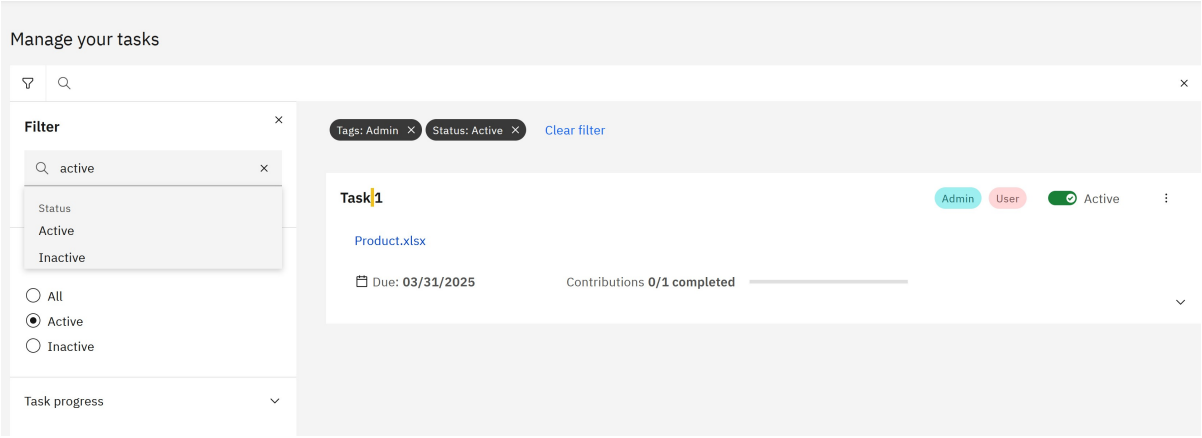

Use one of the following methods to filter on items:

- Type a keyword in the search field.

  The search results appear below the search field. Click the result that you want to filter on. The items that contain the search result are expanded in the **Filter** panel. Select the related check box or button, and click **Apply**. The filter tag is added in the content panel, and the tasks or sections that contain the filtered results are expanded.

- Expand the items in the panel.

  Select the related check box or button, and click **Apply**. The filter tag is added in the content panel, and the tasks or sections that contain the filtered results are expanded.




For both tasks and sections, filter and search results are not saved when you reload a page. The results are saved only when you refresh a plan.

For more information, see Search and filter in applications and Search and filter in plans.

# What's new in modeling - 2.0.102

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.102.

## Websheet enhancements

Learn about the enhancements that have been made to Websheets in the IBM Planning Analytics Workspace modelling workbench.

**Note:** The following features require TM1 11.8.29 or later, or TM1 v12.

**Make a websheet public or private**

You can now set a websheet to be public or private. A private websheet can only be viewed or edited by you.

To make a public websheet private, complete the following:

1. Right-click on a public websheet in the database tree.
2. Click **Security**.
3. Click **Make private**. The folder icon changes to indicate that it is now a private folder .

To make a private websheet public, complete the following:

1. Right-click on a private websheet in the database tree.
2. Click **Security**.
3. Click **Make public**. The folder icon changes to indicate that it is now a public folder .

**Manage websheet security**

In addition to setting a websheet to be public or private, you can now modify the security of websheets and the folder structure used to store them.

To manage the security of a websheet, complete the following:

1. Right-click on a websheet in the database tree.
2. Click **Websheet security**. The `PlanSampLD:)ApplicationSecurity` tab opens. Each public file that is part of the websheet is displayed in table format.

3. Right-click on the cell that corresponds to the file you want to change the security of. Choose one of the following options:
    a. None: Members of the user group cannot see or edit the file.
    b. Read: Members of the user group can see the file, but cannot edit it.
    c. Write: Members of the user group can read and edit the file.

**Rename websheets and folders**

You can now rename websheet and folders.

To rename a websheet or folder, complete the following:

1. Right-click on the websheet or folder in the database tree.
2. Click **Rename websheet** or **Rename folder**, depending on the file you selected to rename. A dialog box appears.
3. In **New websheet name** or **New folder name**, type the new name of the websheet or folder.
4. Click **Rename**.

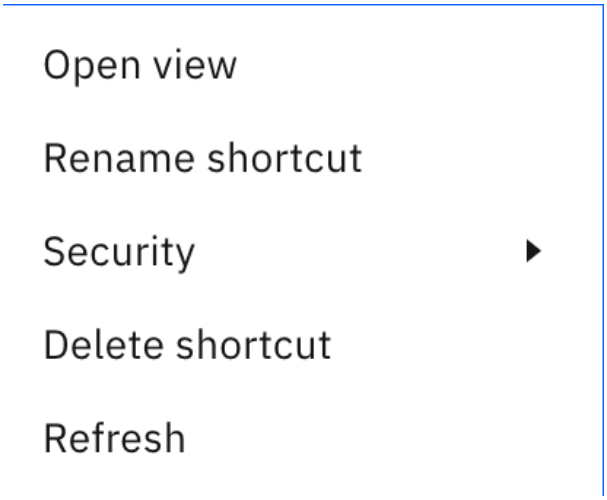
**Manage native view shortcuts**

You can now add and manage native view shortcuts in websheet folders from the database tree.

To add a native view shortcut, complete the following:

1. Right-click on a native view and select **Add shortcut**. The **Add shortcut** menu appears.
2. Type the name of the shortcut in the **Name of shortcut** section.
3. Choose a destination for the shortcut.
4. Click **Add**.

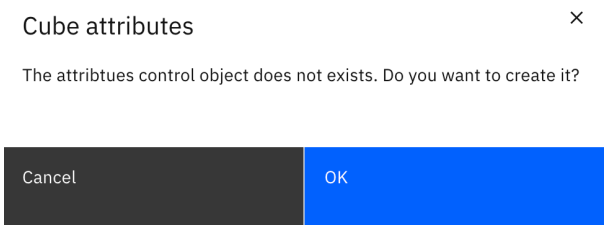
You can right-click on a view shortcut to see additional options for managing it:




## Missing attributes control cubes

If you attempt to edit or view attributes for cubes, dimensions, processes. or chores, and the required attributes control cube is not present in your environment, a message informs you that the control cube is absent.

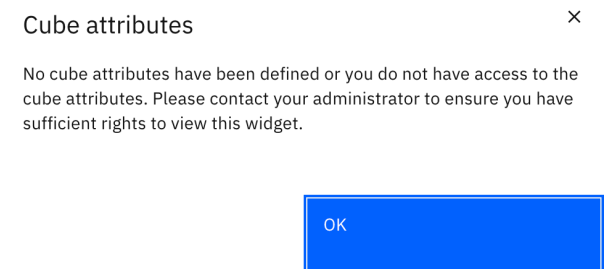
If you are an administrator on the TM1 database, the message prompts you to create the control cube:




Click **OK** to create the relevant attributes control cube and view attributes:

- }CubeAttributes
- }DimensionAttributes
- }ProcessAttributes
- }ChoreAttributes

If you are not an administrator on the TM1 database, the message is as follows:

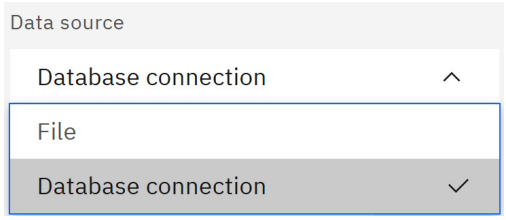



## Import data or members from an ODBC source

You can now import data into a cube or members into a dimension using an ODBC data source. Previous version of Planning Analytics Workspace supported import only from a delimited text file.

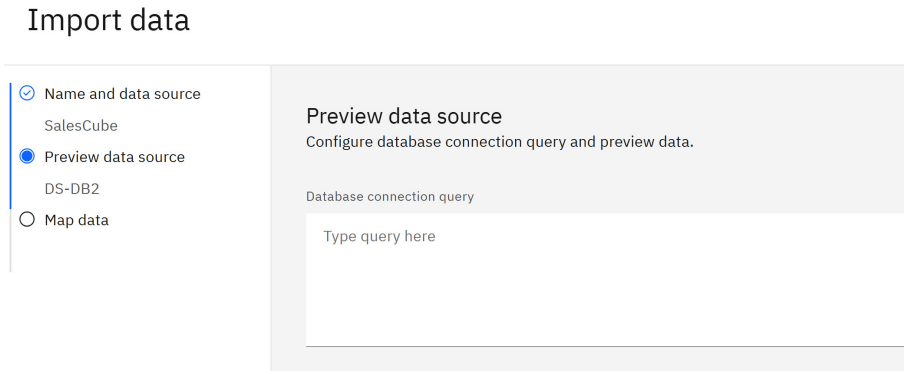
**Note:** TM1 Database 12 requires the ODBC Data Connector to use a database connection for a data source.

When you initiate an import, you'll see a new **Database connection** option on the **Data source** menu.




When you select **Database connection**, you are prompted to select a connection and optionally provide authentication credentials. If the database supports Unicode, enable the **Unicode** option.

You can then specify a query to extract information from the database.




After you establish a connection and execute a query, the procedure to import members into a dimension is the same as for a delimited text file. See Import members for more information.

After you establish a connection and execute a query, the procedure to import data into a cube is the same as for a delimited text file. SeeImport data for more information.

# 2.0.101 - What's new, February 18, 2025

Learn about new features and known issues in version 2.0.101 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.101

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.101.

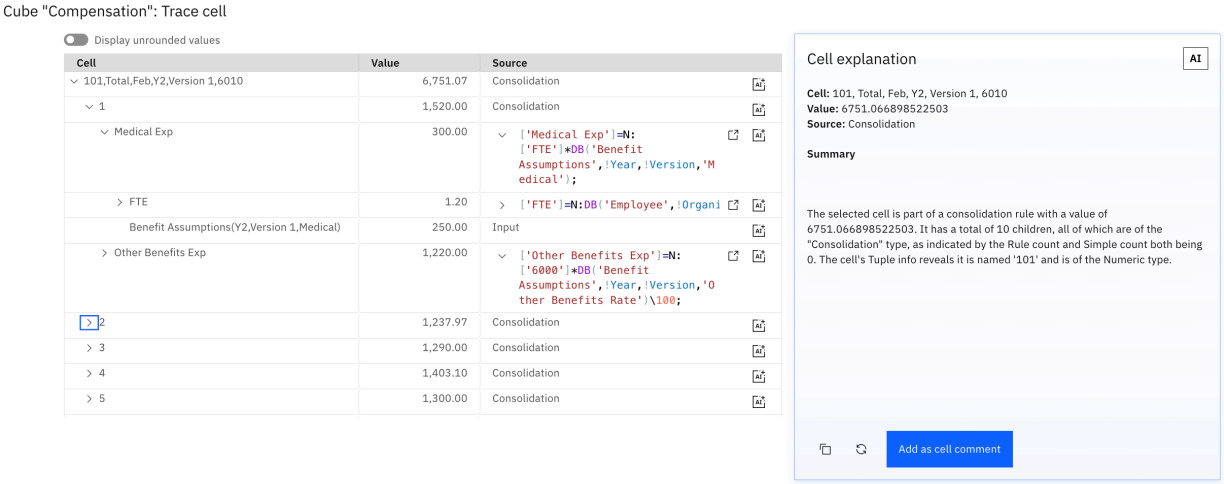
### Use AI to generate cell explanations

Use the new **Explain cell** feature to get an explanation of the underlying values that contribute to a specific cell value. Cell values can be a result of complex calculations, several consolidations, or rules. The **Explain cell** feature uses generative AI to summarize the cell intersection, underlying logic of the cell value, and source type.

The initial release of the **Explain cell** feature is available only in Planning Analytics Workspace Cloud and in English only.

**Important:** The **Explain cell** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM® sales representative or go directly to your IBM account.

To get a cell explanation, right-click a cell in a cube view. Select **Explain cell [AI]** to generate the explanation.




The generated explanation includes a table, with the following columns, and an AI-generated summary:

**Cell**
: **Cell** identifies the original cell by its coordinates in the cube, by using a comma-separated list of member names. If there are members that contribute to the original value, you can drill down to view those members, their values, and source.

**Value**
: Displays the value of the original cell or member that contributes to the value of the original cell. If the value is over 10,000, the value that is displayed is truncated to an integer, otherwise the value shows two decimal places. You can hover over the displayed value to view the actual raw value as it exists in the TM1 database. To view the actual values for all data, enable **Display unrounded values**.

**Source**
: Identifies the source of the value: Consolidation, Input (a leaf cell that is not rule calculated), or rule. If the source is a rule, the entire rule is displayed. If the rule is too long to appear in its entirety within the confines of the column, click **Expand row** to reveal the full rule. You can also click the **AI** icon in each source row to generate an explanation for that specific cell.

The **Explain cell** feature uses granite-3-8b-instruct from the Granite family of IBM foundation models to generate a descriptive summarization of the cell intersections.

After you generate a cell explanation, you can add the summary to the related cell as a comment.

For more information, see Get cell explanation with AI.

## Use the Planning Analytics AI assistant to generate chart insights or impact analysis

The Planning Analytics AI assistant now accepts commands to generate chart insights for a visualization or impact analysis for an exploration.

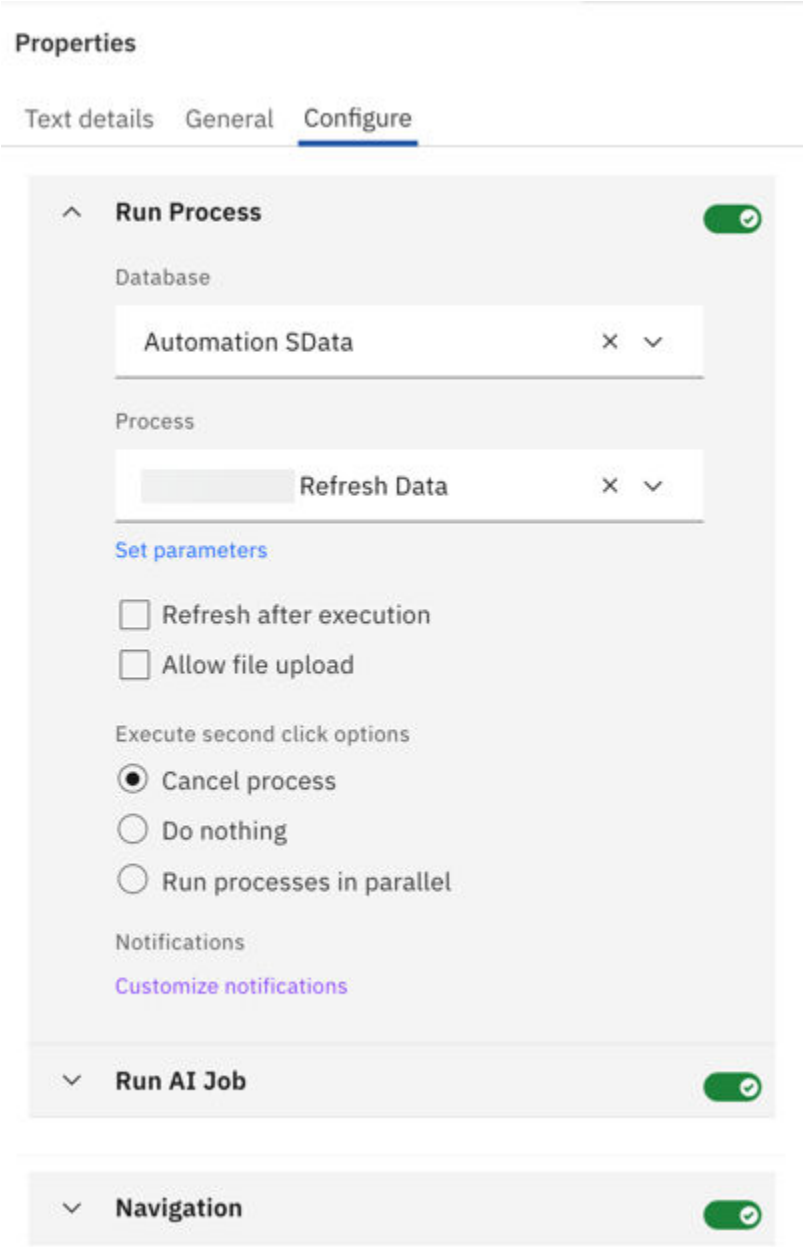
To generate chart insights, select a visualization in a book and then open the Planning Analytics AI assistant. Enter a command such as `Show insights` or `Explain visualization` to display AI-generated insights in the form of text summaries and key points to help you interpret the data in the visualization.

To generate impact analysis, select an exploration in a book and then open the Planning Analytics AI assistant. Enter a command such as `Impact analysis`, `Analyze view`, or simply `Analyze` to view an AI-generated analysis that explains the interaction of data fields in the exploration. The analysis also identifies and quantifies relationships across data in the exploration.

## New design for the Action Button configuration options

You can now easily find all Action Button configurations (Run Process, Run AI Job, and Navigation) on the new **Configure** tab under **Properties**. Previously, each configuration had its own tab under **Properties**.

Simply expand each configuration section to see the available options.



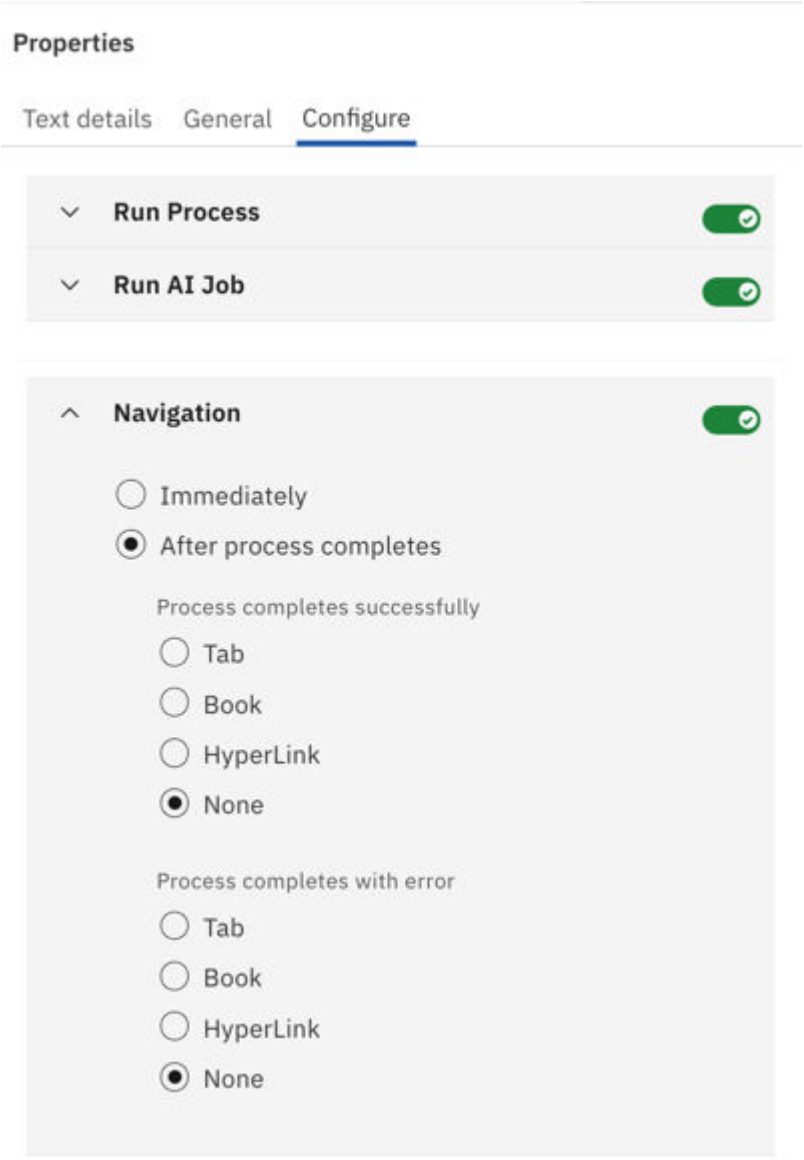

The **General** tab still contains the **Appearance** and **Layout** options for action buttons.

Click inside an Action Button in **Edit** mode to view the **Text details** tab under **Properties**.

## Advanced Action Button configuration for TurboIntegrator processes and navigation

Planning Analytics Workspace now supports advanced configurations of TurboIntegrator processes and navigation between books or tabs in a book. You can run a process, an AI job, or both from an Action Button, and navigate immediately or navigate after the process completes.

You can configure these settings from the properties of an Action Button (**Properties > Configure**). Enable any process to see the available settings.




You can configure each process and navigation to run on its own or in any combination with each other. For example, you can configure:

- a process to run and then navigate
- an AI Job to run after a process completes successfully
- an AI Job to run and navigate immediately
- a process to run and after successful completion, run an AI Job and navigation at the same time
- a process and an AI Job to run and then navigate after their successful completion

If you want to navigate after a process completes, you can even specify where you want to navigate to if the process fails.

To navigate immediately upon clicking the Action Button, choose **Immediately** and select the destination.

1. To go to a tab in the current book, click **Tab**, then select the tab that you want to open from the list of available tabs.
2. To go to a tab in a different book, click **Book** and select a book, and the tab you want to open. Click **Pass context** to pass context for any items that have synchronization that is enabled in both the current tab and the destination tab.

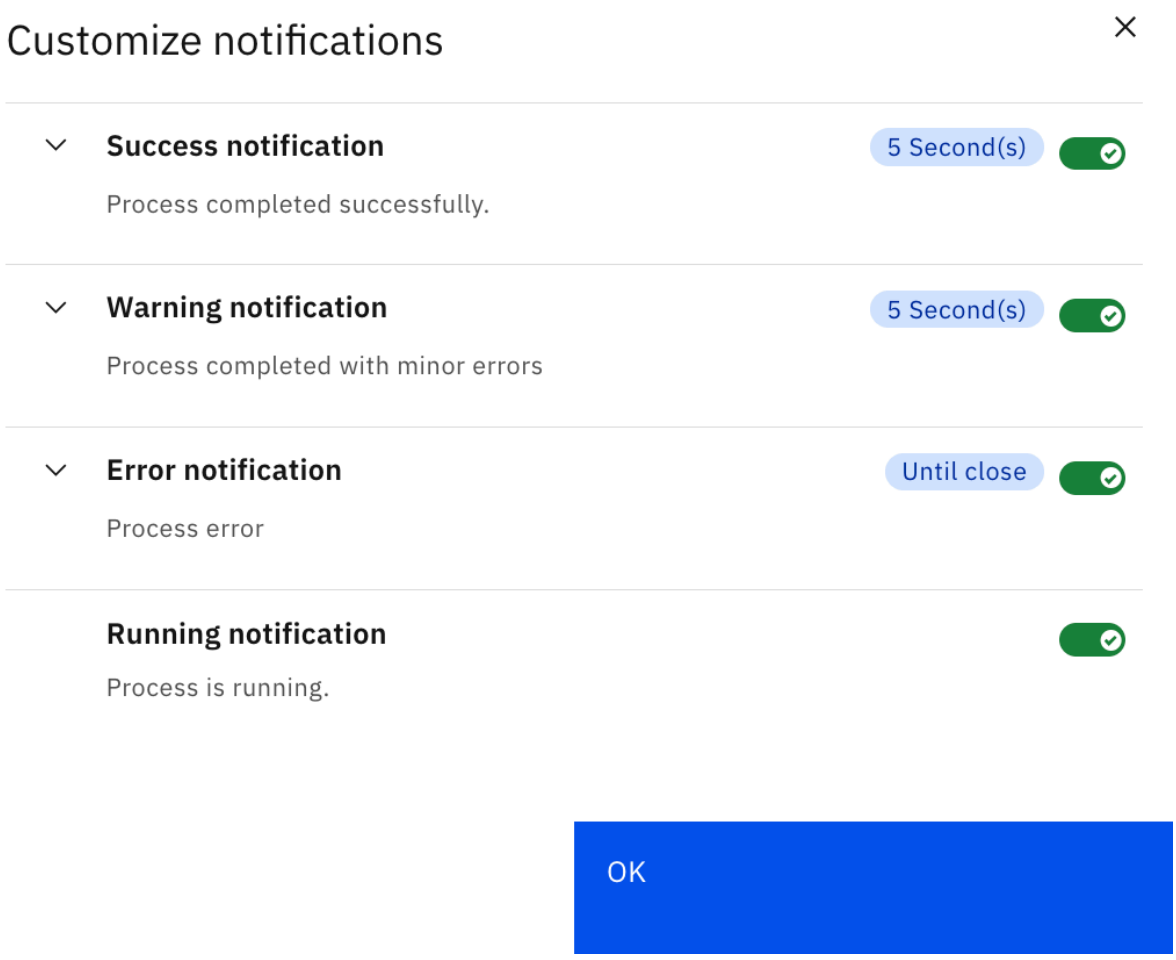
   If you configure an Action Button to navigate to a tab in another book, you can rename the target book and/or sheet without having to update the button properties. The button automatically resolves any name changes to target tabs or books.
3. To open a URL, click **Hyperlink**, then enter a fully qualified URL.

To navigate after a process completes, select **After process completes** and select where you want to navigate to when the process completes successfully and when the process completes with errors.

## Implement custom notifications for TurboIntegrator processes

You can now configure custom notifications to display when a TurboIntegrator process is run from an action button to display its status. Create a notification to display when a process completes successfully, completes with a warning, or fails.

You can also choose the amount of time the notification displays on the screen or if the notification should display until the user closes it manually.




To configure custom notifications for TurboIntegrator processes, follow these steps:

1. In **Edit** mode, select the action button that runs the process for which you want to customize notifications.

2. Go to **Properties > Configure > Run Process** and click **Customize notifications**.
3. By default, all notification types are enabled. You can turn off the notifications, which you don't want to display.
4. Expand the notification type that you want to customize.

   **Note:** The **Running notification** message can't be customized. However, you can turn it off if you don't want this notification to display.
5. Enter the notification message and select the amount of time it displays on the screen.

   **Note:** Select **Until close** for **Time on screen** if you want the user to close the notification manually.
6. Click **OK** to save your settings.

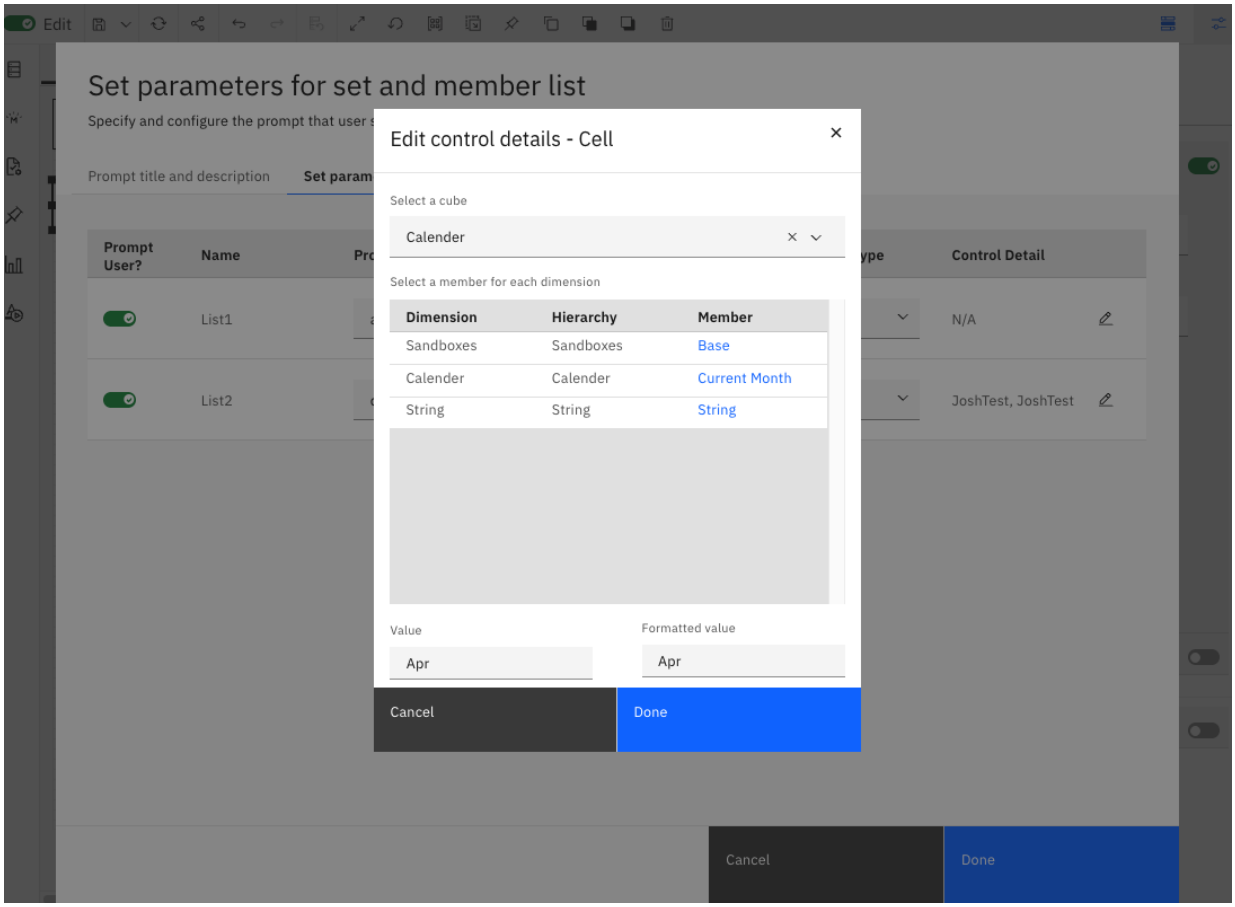
## Add cell values as parameters in TurboIntegrator processes that are run from an Action Button

You can now pass cell values as parameters in processes that run from an Action Button. You can configure the Action Button to refer to a specific cell value, that is string or numeric, by selecting a cube and the members that determine the intersection.

To add cell values as process parameters:

1. In edit mode, select the button you want to configure.
2. Go to **Properties**, enable **Run Process**, and select the database and process.
3. Click **Set parameters** and specify the prompt, then:
   a. Select **Cell** as the **Control type**.
   b. Click **Edit** and select a cube and the member for the cell value intersection.

      **Note:** The parameter type and the cell value type must match. For example, if the parameter type is numeric, the specified cell must also be a numeric value.

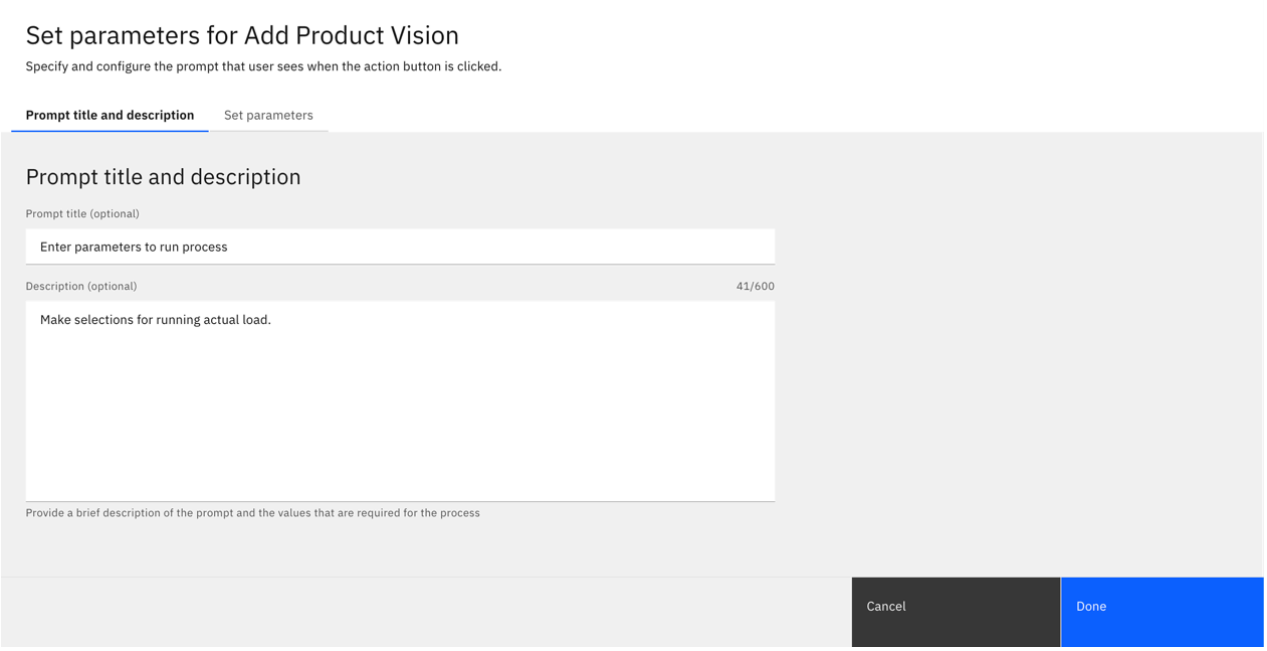



If you change the cell value in a cube viewer or through a single cell widget, the updated cell value reflects in the process parameters.

## Customize action button parameter title and description

You can now customize the Action Button parameter title and add a description for a process that requires parameter values. When the process is run, the prompt dialog displays the title and description you

created. Use this new feature to provide contextual information about the process or instructions for setting parameters.




To customize the parameter title and add a description for an Action Button process, follow these steps:

1. In **Edit** mode, select the Action Button which runs the process that requires parameter values.
2. Go to **Properties > Configure > Run Process** and click **Set parameters**.
3. Enter the prompt title and a description and click **Done**.

When you run the process now, the prompt dialog displays the new title and description.

## Customize cell color based on data state

The **Color cells by data state** property for views was enhanced to allow customization of cell color based on its data state. You can pick a custom background color for rule-derived cells and read-only cells, and even customize the color of sandbox values. Additionally, the **Color cells by data state** property has moved and can now be found in the **Format manager**, under **Data area**.

**Important:** The **Color cells by data state** property overrides any other background color set for the cell.

To enable the **Color cells by data state** property for a view and customize the background color of cells, follow these steps:

1. Select a view and click **Format manager** in the toolbar.
2. Go to the **Data area** on the **View** tab.
3. Under **Table styles**, select **Color cells by data state** and choose the colors you want for **Rule-derived background**, **Read-only background**, and **Sandbox font**.
4. Click **Apply**.

**Note:** Rule-derived cells include cells with rule-derived values, calculations, and set consolidations. The **Color cells by data state** property does not apply to consolidated members.

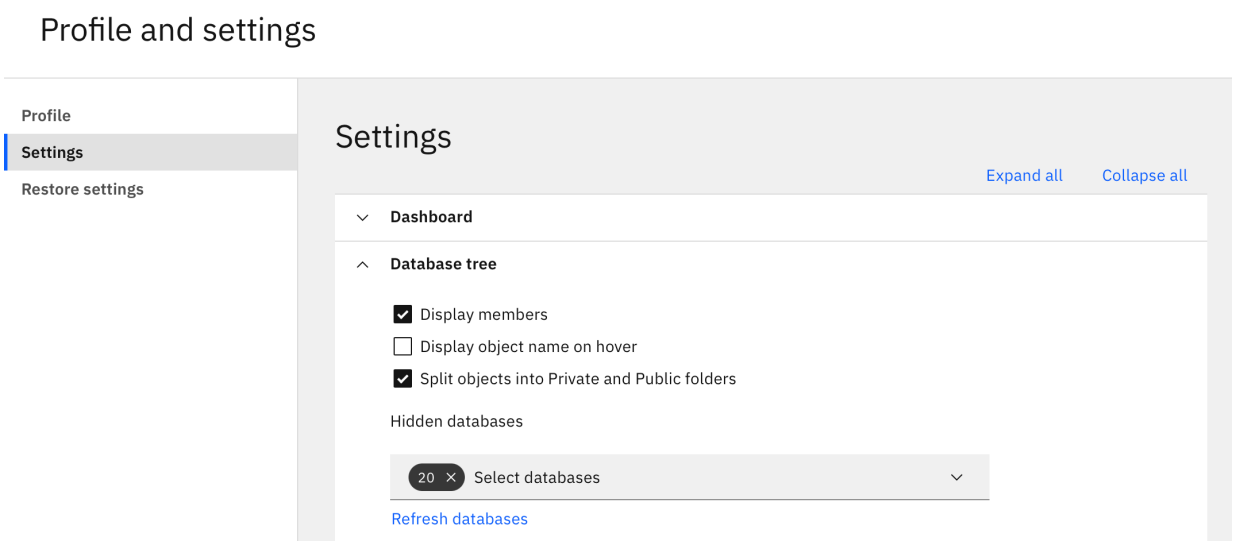
Previously, when the **Color cells by data state** property was enabled, read-only cells displayed in gray, calculated and rule-derived cells displayed in green, and sandbox values in blue font. These were the default settings and could not be changed.

## New sorting options for views and sets in the Database tree

A new folder structure in the Database tree now lets you sort views and sets into public and private folders. This feature is available in books and the modeling workbench.

To display the public and private folders in the Database tree, enable **Split objects into Private and Public folders** in the database tree settings.

1. In the **Personal menu**, click **Profile and settings**.
2. Click **Settings** and under **Database tree**, select **Split objects into Private and Public folders**.



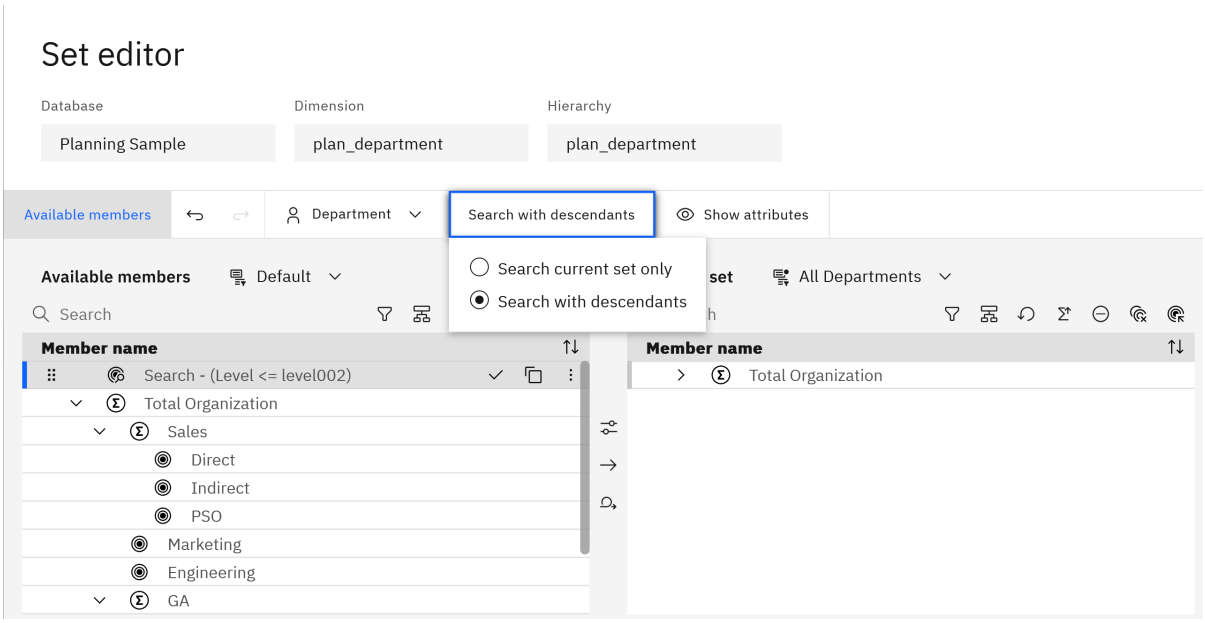

3. Click **Save**.

You can now see the private and public folders in the Database tree and sort views and sets into these folders.

## Change the search scope in the set editor

The set editor now includes options to configure the search so that it returns only the current set members, or the current set members and their descendants.

To support this new behavior, the **Search with descendants** and **Search current set only** settings were added to the set editor.



The new search settings affect both the search and filter results in sets that include consolidated members. Search and filter in the **Available members** and **Current set** panels are affected in the same way.

To include the descendants of members when searching or filtering, select the **Search with descendants** option.

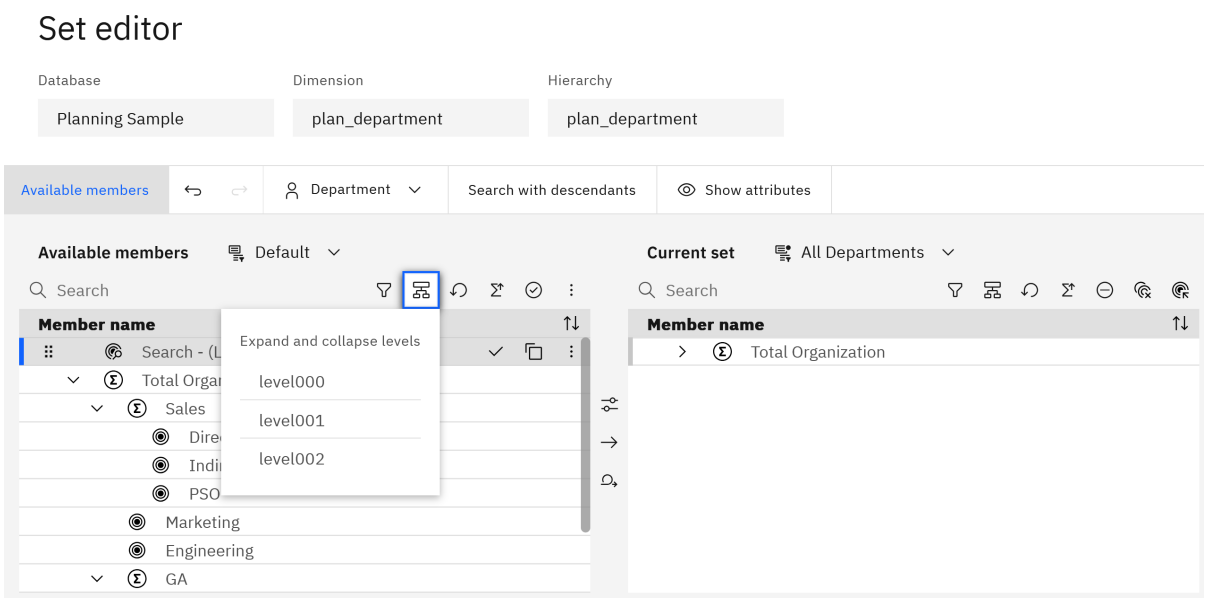
To search or filter only the current set members, select the **Search current set only** option. This is the default option.

For more information, see Searching in sets and Filtering sets.

## Expand and collapse levels in the set editor

The set editor now includes a button that allows you to expand or collapse all consolidations at once, up to the selected level.

The new **Expand and collapse levels** button is available in the **Available members** and **Current set** panels.




For example, you can use this button to collapse all members and then expand only a specific consolidation to examine the consolidation path.

For more information, see Narrowing down selections in the set editor.

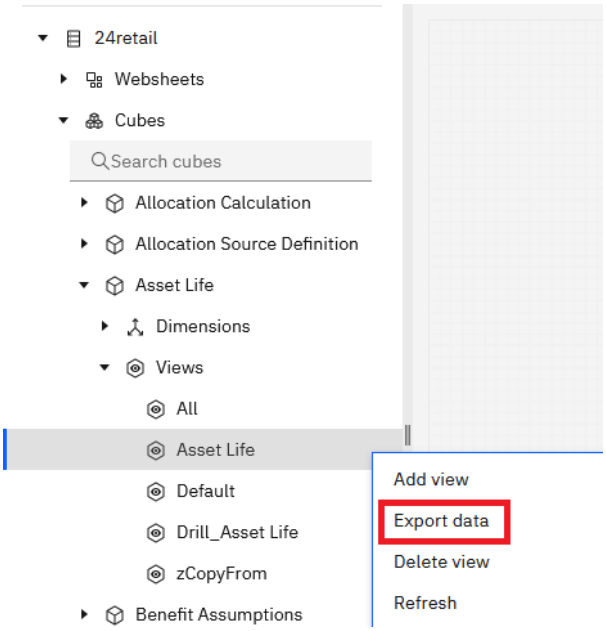
## Export options in books

The export options for a cube and cube view, and for a dimension and dimension hierarchy, are now available from a book in the same way as they are available from the modeling workbench. The exported `.csv` file is saved to your local `Downloads` directory.

### Export the cube or view data from a book

To export data from a cube, right-click the cube in a book **Databases** tree, and then click **Export data**.

To export data from a view, right-click the view in a book **Databases** tree, and then click **Import/Export** > **Export data**.



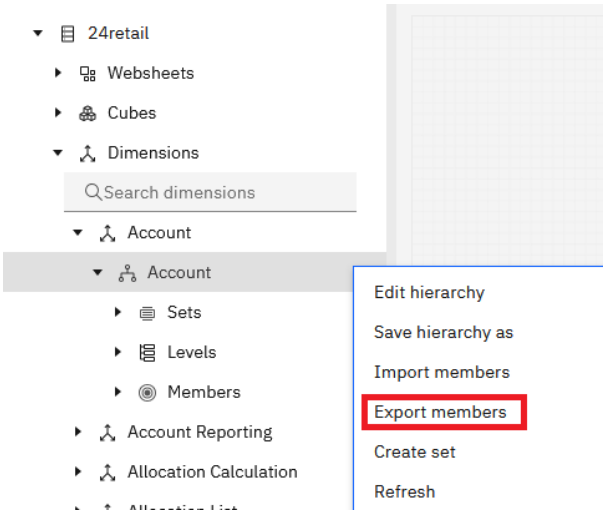

When you export data, you can set options that determine:

- The set and/or members to include in the export
- A range of numeric or string values to include in the export
- Whether consolidated, rule-derived, or zero value items are included in the export

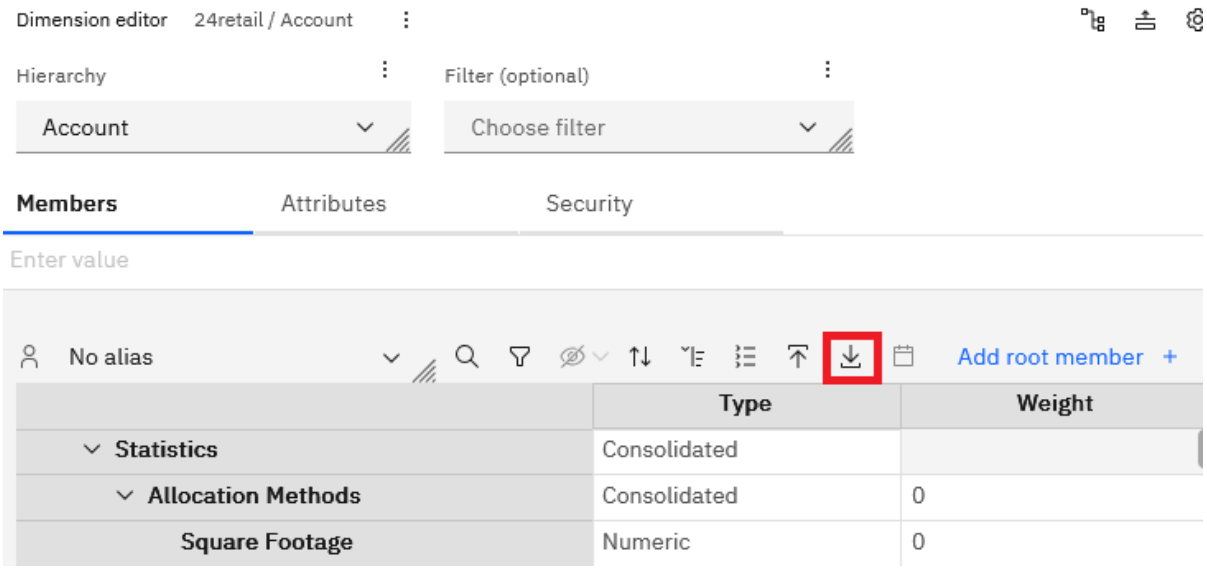
For more information, see Export data from a cube or view.

## Export a dimension or hierarchy from a book

To export from a dimension in a book, right-click a dimension or hierarchy in the **Databases** tree, and then click **Export members**.




You can also click the **Export members** icon on the dimension editor to export a dimension.




When you export a dimension or hierarchy, the resulting `.csv` file includes the following information about each member:

- Name
- Parent-child relationship
- Type
- Weight
- Defined attribute names and values

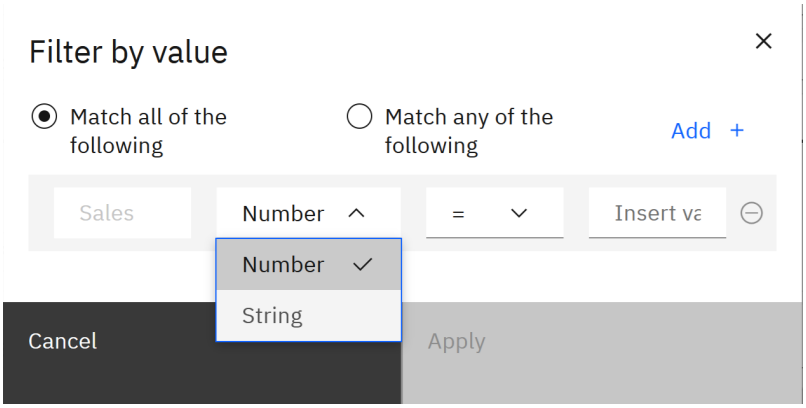
  Attribute names are prefixed with either ATTR_N or ATTR_S when they are exported, depending on the attribute type (numeric or string, respectively).

For more information, see Export a dimension or hierarchy.

## Create filters based on string values

The **Filter by value** filter in the cube viewer now supports string values, in addition to the previously supported numeric values.

To use the new filter option, right-click a row or column header, select **Filter**, and then select **Filter by value**. From the drop-down menu, select **String**.

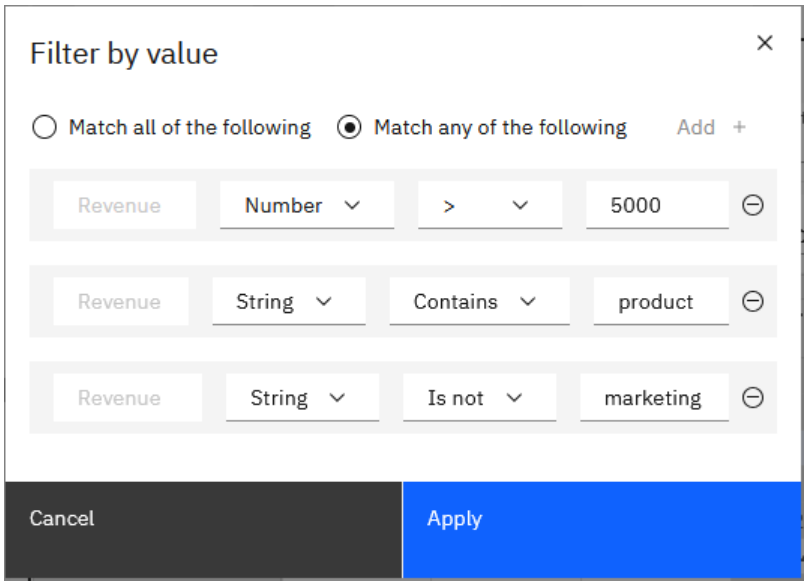



The following operators are available for string-based filters:

- Is (equal to)
- Is not (is not equal to)
- Contains

- Starts with
- Ends with

You can create filters with mixed numeric and string data, as shown in the following example:




The filter icon is applied to a row or column for which the filter is created.

For more information, see Filter rows and columns based on cell value.

## Manage data entry and data spreading in consolidated cells

You can now allow or prevent data spreading when values are entered in consolidated cells in an exploration.

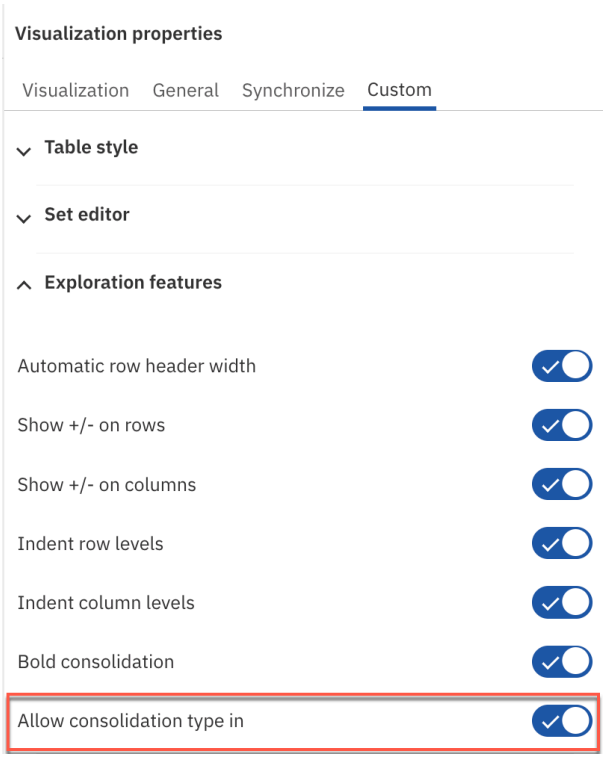
When you type data directly into a consolidated cell, a proportional data spread is performed by the Planning Analytics database. To prevent this type of spreading from happening accidentally, an option to allow or prevent the ability to enter values directly in consolidated cells is now available. This option is set as a property of an individual exploration; if a book contains multiple explorations, the property can be set independently for each exploration.

To manage the ability to enter data directly in consolidated cells of an exploration:

1. Open the book containing the exploration in **Edit mode**.
2. Click the exploration.
3. Click **Properties**.
4. Click the **Custom** tab.
5. Expand **Exploration features**.
6. Toggle the **Allow consolidation type in** option.

   When the option is enabled, users can enter a value directly in a consolidated cell, initiating a proportional data spread.

   When the option is disabled, users cannot enter values in consolidated cells.




Regardless of the **Allow consolidation type in** setting, you can right-click any consolidated cell in an exploration and then click **Spread data options** to apply data spreading.

## Support for parameters in the IBM Planning Analytics Workspace MDX editor

You can now parametrize MDX queries in IBM Planning Analytics Workspace.

The default MDX generation uses a query state, which consists of the sets for each hierarchy, member selections for hierarchies on titles, and any applied operations (for example, expand, collapse, sort, etc). The query state generates an MDX query to retrieve the corresponding axis and cell data from the TM1 database.

A parameterized MDX involves the use of an MDX template. MDX templates allow you to override how the query state is used to generate the MDX. An MDX template contains the following:

- Variables that can be substituted during MDX generation.
- Variable declarations which define the members or sets in the query state which are substituted for the variables in the MDX template.

Supporting parameters allows for more control over the MDX, while allowing the view to respond to the specified member and set synchronization events based on how the template is defined.

**Note:** You can parameterize MDX queries only in books; this capability is not available on a modeling workbench.

The following variables can be defined in an MDX template:

**Set variables**

Set variables define a reference to a set for a hierarchy defined in the query state.

For example:

```
{
      "Name": "<0>",
      "HierarchyID": "[plan_time].[plan_time]",
      "Type": "SetVariable"
    }
```

**Slicer member variables**

Slicer member variables define a reference to the selected member for the query state.

For example:

```
{
      "Name": "<2>",
      "HierarchyID": "[plan_version].[plan_version]",
      "Type": "SlicerVariable"
    }
```

**Set member variables**

Set member variables define references to members of a set at a specified index.

For example:

```
{
      "Name": "<1>",
      "HierarchyID": "[plan_chart_of_accounts].[plan_chart_of_accounts]",
      "Type": "SetMemberVariable",
      "Index": 0
    }
```

The following is an example of a complete MDX template:

```
{
  "MdxTemplate": "SELECT {<0>} ON 0, {<1>} ON 1 FROM [plan_BudgetPlan] WHERE (<2>,
[plan_business_unit].[plan_business_unit].[10110])",
  "Variables": [
    {
      "Name": "<0>",
      "HierarchyID": "[plan_time].[plan_time]",
      "Type": "SetVariable"
    },
    {
      "Name": "<1>",
      "HierarchyID": "[plan_chart_of_accounts].[plan_chart_of_accounts]",
      "Type": "SetMemberVariable",
      "Index": 0
    },
    {
      "Name": "<2>",
      "HierarchyID": "[plan_version].[plan_version]",
      "Type": "SlicerVariable"
    }
  ]
}
```

## How to parametrize an MDX query

To parametrize an MDX query, complete the following steps:

1. In the cube viewer, click the edit MDX icon MDX to access the MDX.
2. Click the MDX template variables icon </>. The **MDX parameters** dialog box opens.
3. To add a parameter, click the **Add parameter** button and set the following options:
   a. **Name**: The parameter id. The name can only consist of numeric values.
   b. **Dimension**: The dimension to define in the parameter.
   c. **Hierarchy**: The hierarchy to define in the parameter.
   d. **Type**: The type of variable to define in the parameter. You can choose a **Set**, **Set member**, or **Slicer** variable.
   e. **Index**
4. To add the current parameter to the list and create another parameter, click **Add**. To add the current parameter to the MDX and close the **MDX parameters** dialog box, click **Add and close**.
5. Click **Done**.

### Known limitations

There are a few current known limitations with MDX templates:

- Apart from the defined variables, MDX template views are read-only. They do not respond to events that change the structure of the query (for example, pivoting, expand, collapse, sort, etc).
- MDX template views only respond to member and set synchronization events they are parameterized to respond to.
- Metadata change recovery is only applied to sets and members referenced by variables.

## What's new in plans and applications - 2.0.101

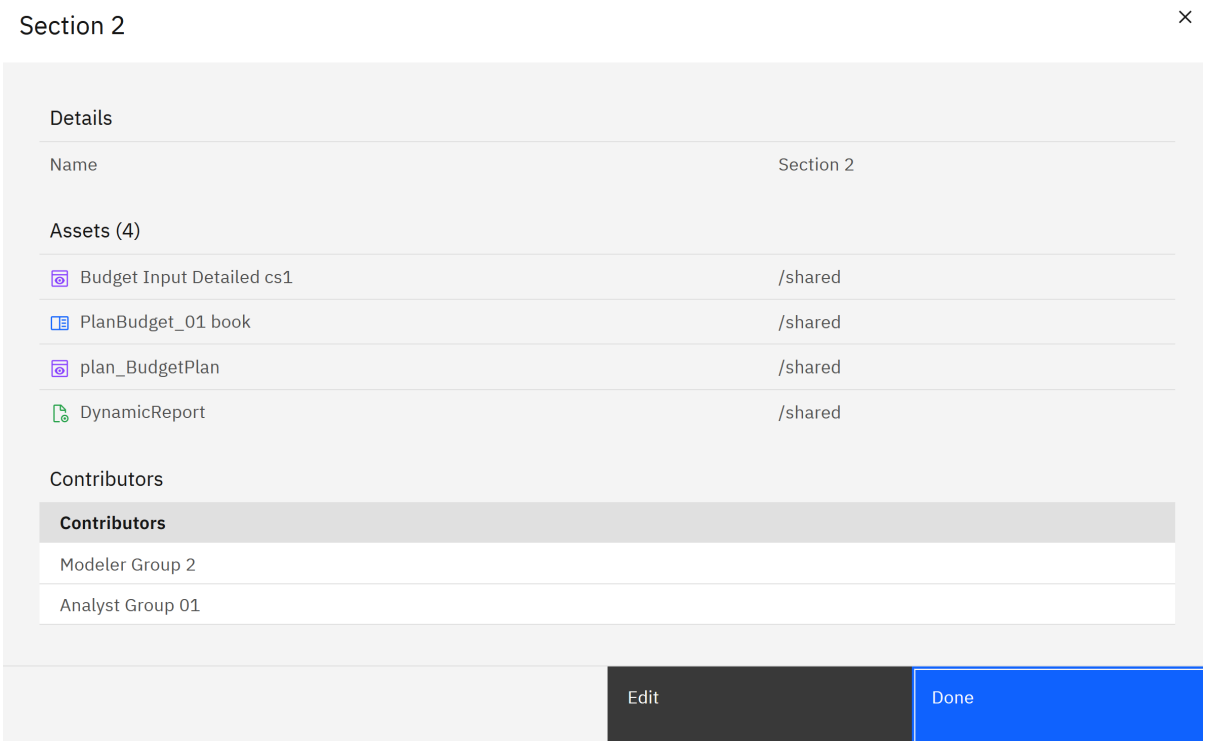
The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.101.

### Summary pages in applications and plans

Sections in applications and tasks in plans now have a summary page.

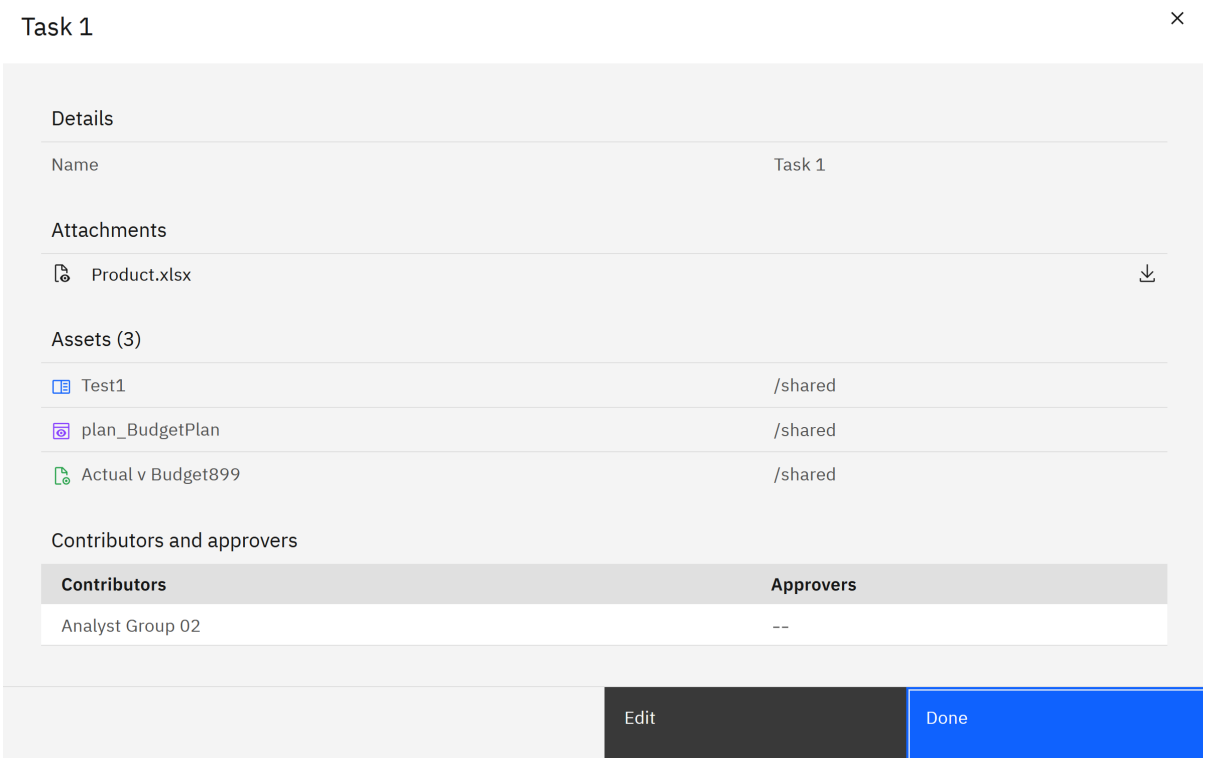
The summary pages provide a compact view of the assets that the section or task contains.

To view the summary for an application, on the **Application sections** tab, expand the section, and click the **Summary** link. A page similar to the following example is displayed.




For more information, see Add sections and contributors to an application.

To view the summary for a plan, on the **Tasks** tab, expand the task, and click the **Summary** link. A page similar to the following example is displayed.



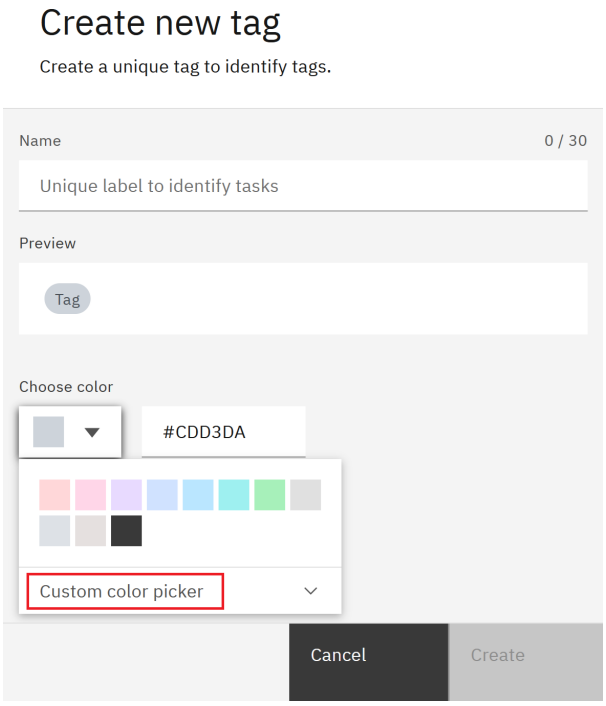

For more information, see Add tasks to a plan and assign contributors and approvers.

## Custom color picker for tags

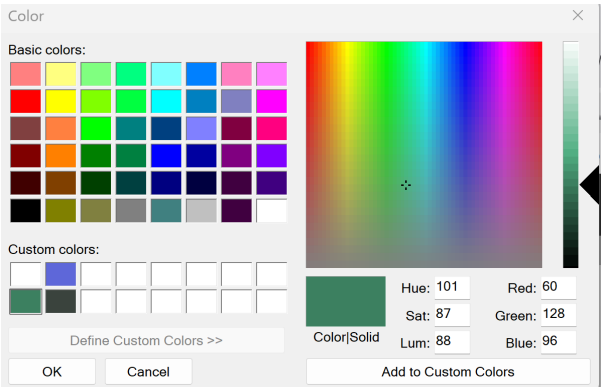
When you create color-coded tags in application sections and plan tasks, you can use the color picker to apply a custom color to your tag.

In previous product versions, only a limited number of colors were available for tags.

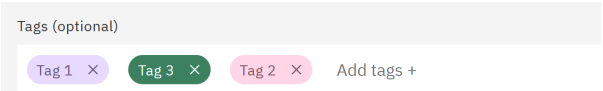
To apply a custom color when creating a tag, click **Custom color picker** on the **Create new tag** dialog box.



The color picker that is specific to your browser is displayed. The following image is from Firefox.




In the color picker, specify the color that you want to apply to the tag.




For more information, see Add sections and contributors to an application and Add tasks to a plan and assign contributors and approvers.

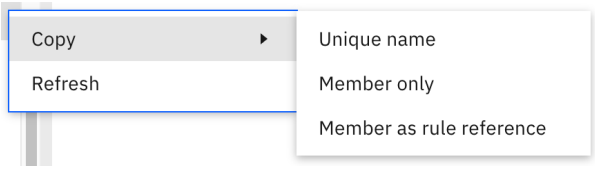
# What's new in modeling - 2.0.101

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.101.

## Copy menu added to the IBM Planning Analytics Workspace modelling workbench

A **Copy** menu has been added to the context menus for objects in the Databases tree,

The **Copy** menu allows you to copy different parts of member and hierarchy scripts to the clipboard, which allows you to more easily edit the MDX, rules, and TurboIntegrator processes.




To access the **Copy** menu, right-click on a member or hierarchy in the tree, click **Copy**, then select one of the following options:

- **Unique name**: Copies the unique name. This is used in the MDX.
- **Member only**: Copies the member. This is used in rules and TurboIntegrator processes.
- **Member as a rule reference**: Copies the member as a rule reference. This is used in rules.

When an option is selected, the corresponding text is copied to the clipboard.

## Generate pick lists for selected cells in the cube viewer

A user in the modeling workbench can now create pick lists in the cube viewer by selecting one or more cells and generating a pick list for them. This is a simpler method of setting up pick lists comparing to the previously used (and still supported) methods.

The users are guided to create the pick list. The type of pick list that the user can generate depends on the type of cell that the pick list is created in. For numeric cells, the pick list is typically a range of numbers that the user can select from. For string cells, the pick list is a static list and the user can quickly enter

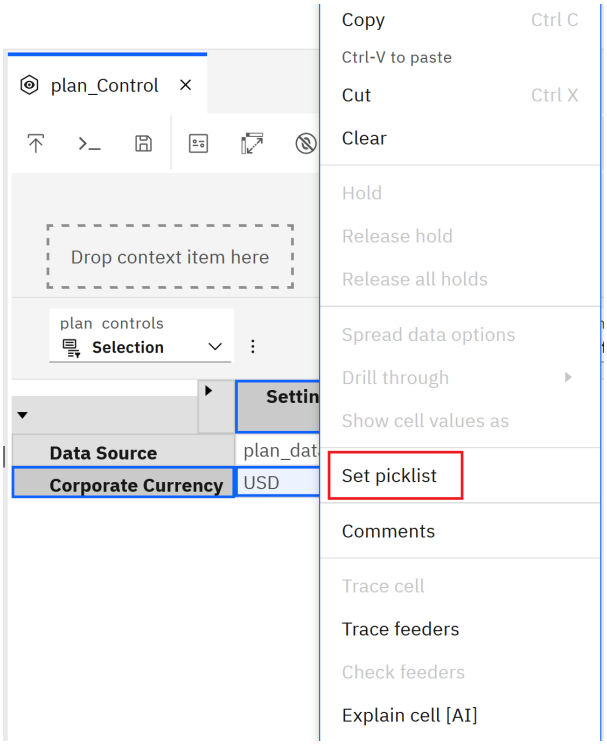
and edit the list of items. For hierarchies and subsets, the user can select from an existing dimension and hierarchy.

Follow these steps to create this new type of pick lists:

1. In the cube viewer, select a single cell or multiple cells of the same type (string or numeric).

   Right-click the cell, and in the menu locate the **Set picklist** option.

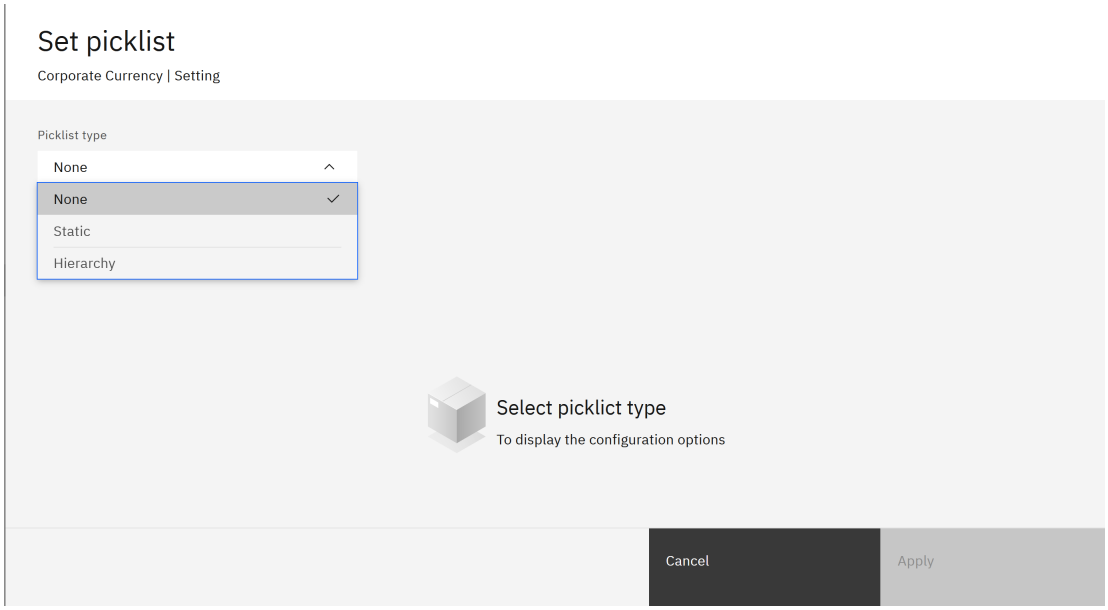
   If this option is not active, the cell cannot be used to create a pick list. Pick lists cannot be created for the following types of cells: rule-based calculations, consolidations, read-only cells, or when the cells are of different types (in a multi-selection).




2. Click the **Set picklist** option.

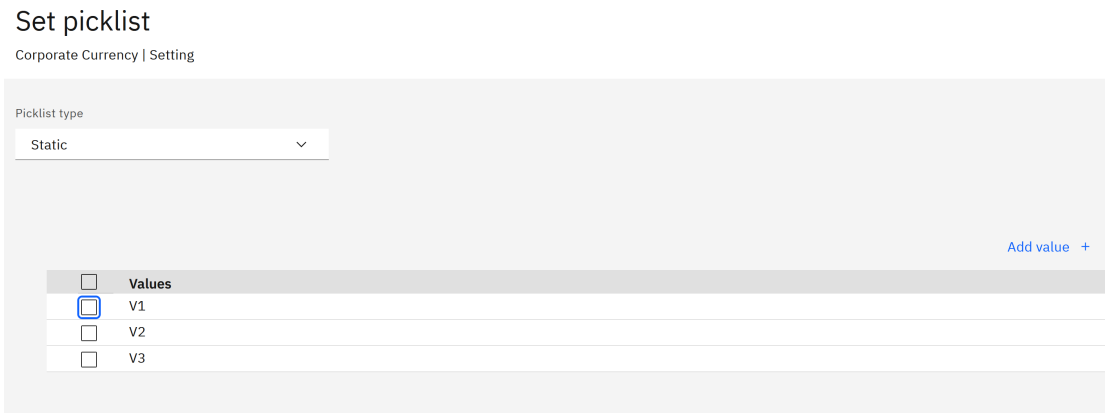
   In the dialog box that appears, select the pick list type to create:

   - None - clear the pick list if it already exists for the selected cell
   - Static - specify pick list values
   - Hierarchy - use all hierarchy members, or a subset of members, as pick list values

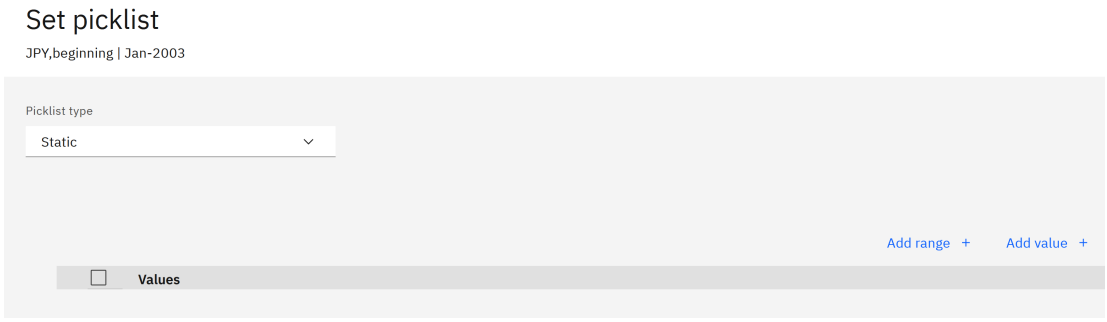



3. If you select the **Static** type, the dialog box that is displayed depends on the cell data type.

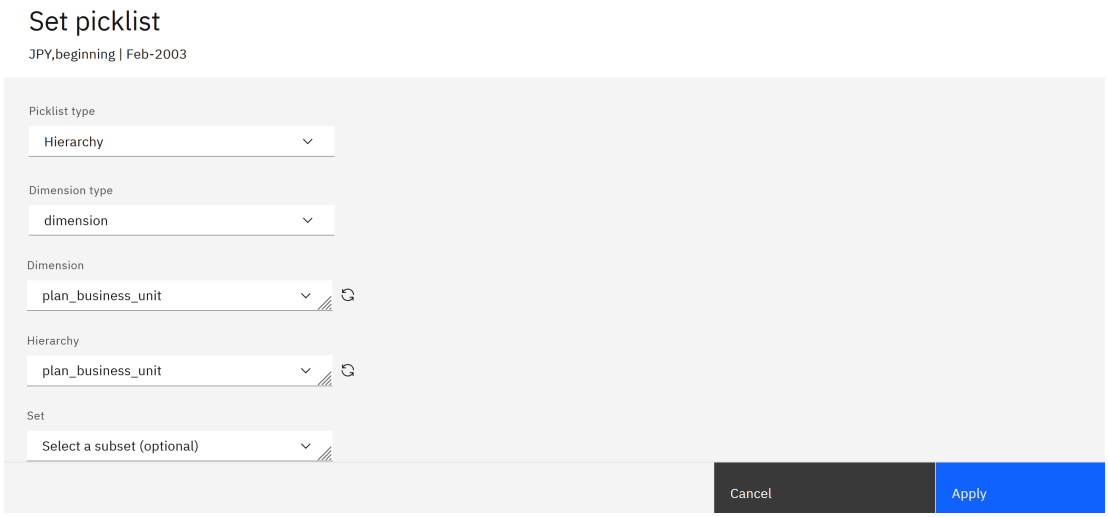
   The following dialog box is displayed for string cells. Click **Add value** to specify the pick list values.




   The following dialog box is displayed for numeric cells. In this case, you can click **Add value** to type the pick list values, or click **Add range** to specify value ranges.




4. If you select the **Hierarchy** type, the following dialog box is displayed.



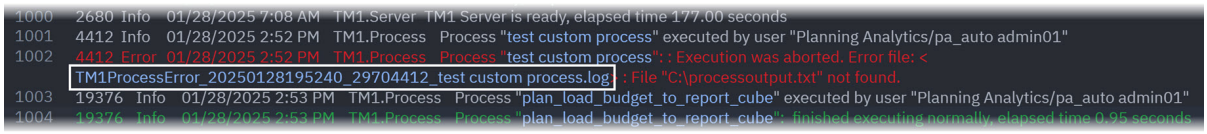

Select a dimension first, and then a hierarchy. The dimension type has two options, **Dimension** and **Control dimension**. The **Set** option is just a case of the hierarchy option. In this case, the pick list is tied to a subset.

For more information, see Pick lists.

## Open process error logs directly from the server log

The Planning Analytics server log contains information about process execution. When a process encounters an error and generates a process error log, you can now open the process error log directly from the server log.

To open the process error log, click the log filename in the server log. Process error logs use the file naming convention `TM1ProcessError_`*`timestamp_processname`*`.log`.



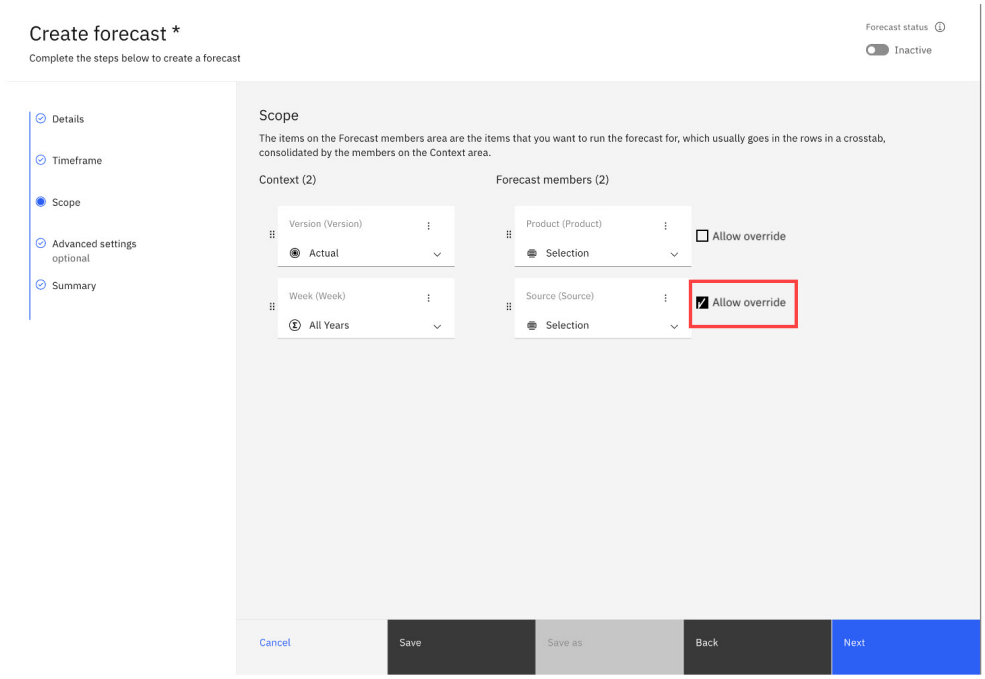

The process error log opens in a new message box. You can review the error log and click to download a copy of the log.

## Increase the flexibility of baseline forecasts by allowing users to override default sets

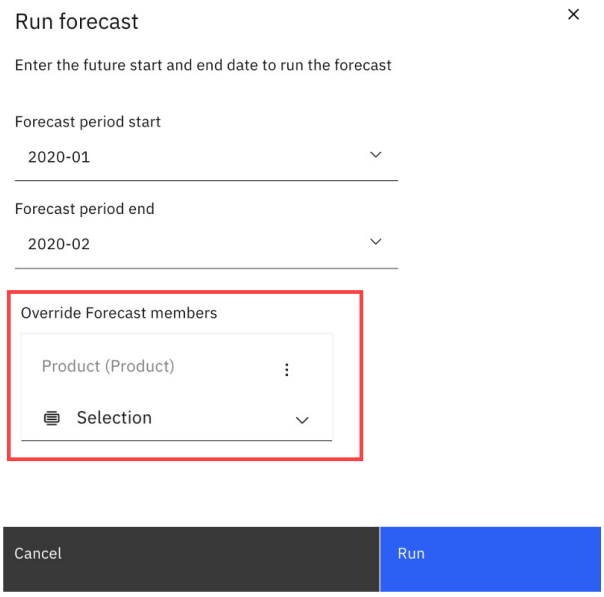
An administrator or modeler can now set an option that enables users to override default forecast members and specify a custom scope when running a baseline forecast.

This makes baseline forecasts more flexible for users and simplifies maintenance for baseline forecast creators, who no longer need to create and maintain multiple forecasts with similar but varying scopes.

To enable users to override default forecast members and specify a custom scope, enable the **Allow override** option when you define the scope of a baseline forecast.




When a user runs a baseline forecast with the **Allow override** option enable, they can use the **Override Forecast members** option to pick an existing set or level for the forecast or open the set editor and define a new set.




Though the user specifies a set or level that differs from the default defined in the baseline forecast, the forecast definition remains unchanged. The custom selection is applied only at run time.

## Control logging levels (TM1 Database 12 only)

You can now specify logging levels for individual loggers in Planning Analytics Administration on TM1 Database 12.

### About this task

The available logging levels are:

**Off**
No logging.

**Debug**
Detailed, technical messages that are useful when customer support or engineering needs to debug the database.

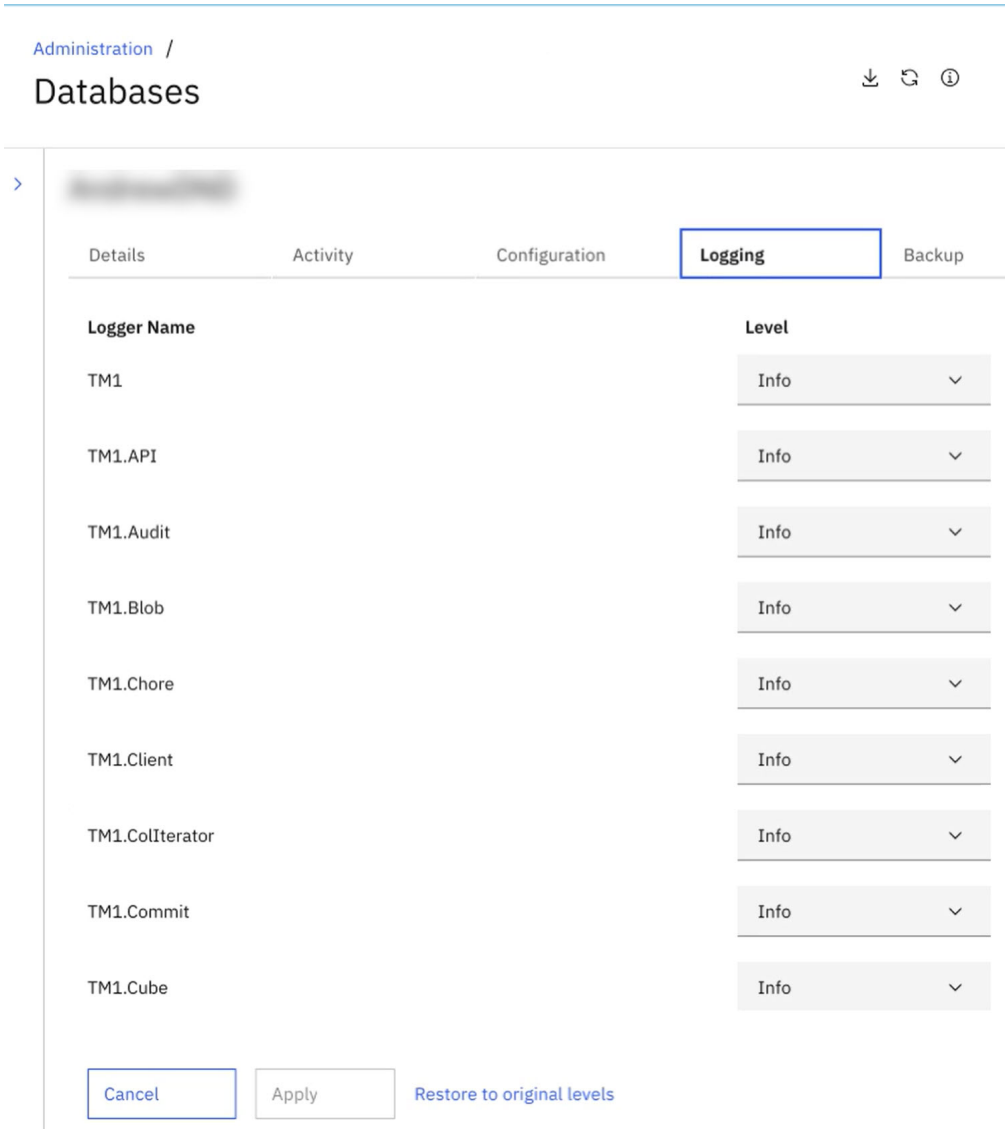
**Info**
Informational messages that highlight the progress of the database and report normal actions within the database.

**Error**
An error condition of which you should be aware. Action should be taken to fix or report the issue to customer support.

The default logging level is Info.

## Procedure

1. Open Planning Analytics Administration.
2. Click the database for which you want to set logging levels.
3. Click the **Logging** tab.




4. To set the level for a logger, click the **Level** menu and select the desired logging level.

The available loggers are listed alphabetically and hierarchically. If you set a logging level for a parent logger before setting a logging level for any child loggers, all child loggers initially inherit the same level, but can be modified independent of the parent. If you set a logging level for a child prior to setting a logging level for a parent logger, the child logging level remains unchanged when you set the parent level.

For example, the TM1 logger is the parent of all other available loggers. Similarly, the `TM1.Event` logger is the parent of the `TM1.Event.Process` and `TM1.Event.Process` loggers.

5. Click **Apply** after you've set all desired logging levels.

# What's new in administration - 2.0.101

The following new and changed administration features are available in Planning Analytics Workspace 2.0.101.

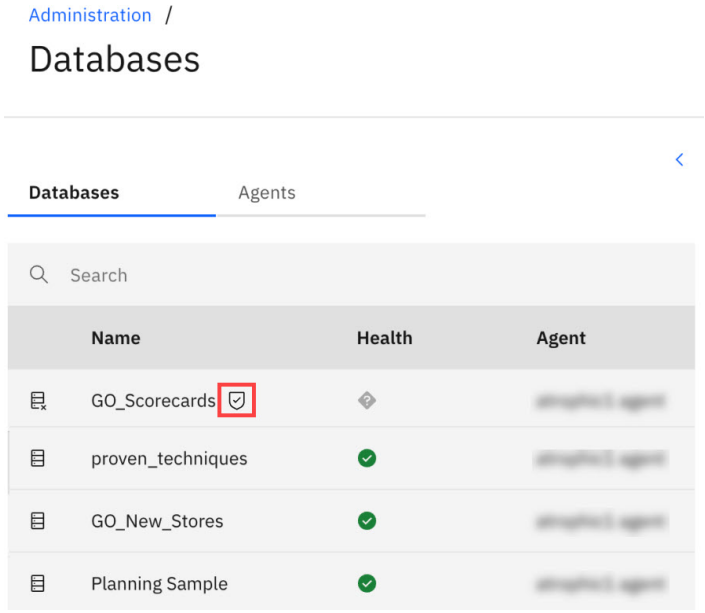
## Identify the database used for authentication on Planning Analytics Administration Local (Local only)

You can quickly identify the database used for authentication in the list of databases on the Planning Analytics Administration page.

**Important:** For the authentication database to be correctly identified in Planning Analytics Administration, the TM1LoginServerURL configuration parameter for Planning Analytics Workspace Local must be specified with an IP address and port. The IP address must match the address specified by the IPAddressV4 parameter in your `tm1s.cfg` file. The port number must match the port specified by the HttpPortNumber parameter in your `tm1s.cfg` file.

`TM1LoginServerURL=https://`
*`IPAddressV4_from_tm1s.cfg:HttpPortNumber_from_tm1s.cfg`*

The database used for authentication is identified by the icon. The icon is displayed only on Planning Analytics Administration Local and only when Planning Analytics Workspace is configured to authenticate against a specific Planning Analytics database.




Administrators can look for this icon to determine which database is used for authentication so they don't accidentally stop the database or modify it in such a way that prevents users from being able to log in to Planning Analytics Workspace.

## Known issues - 2.0.101

The following are known issues in Planning Analytics Workspace 2.0.101. These issues will be resolved in a future release.

### Invitation email is not sent when a new user is added to Planning Analytics as a Service 2.0.101 on Microsoft Azure

When you add a new user in Planning Analytics Administration within Planning Analytics as a Service on Microsoft Azure, the user is added but an invitation email is not sent to the new user.

The new user can log in and use Planning Analytics, despite not receiving an invitation email.

# 2.0.100 - What's new, December 11, 2024

Learn about new features and known issues in version 2.0.100 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Important upgrade requirement for Planning Analytics Workspace Local 2.0.100

You must backup your Planning Analytics Workspace Local content prior to upgrading to 2.0.100 and then restore your content after the upgrade is complete.

Backup and restoration is required because Planning Analytics Workspace Local 2.0.100 uses a new version of MongoDB database and the Planning Analytics Workspace Local installation process cannot directly update content on the database. MongoDB stores books and other assets that are required by your Planning Analytics Workspace environment.

If you do not backup content before upgrading to 2.0.100 and restore data after upgrading, your content will be missing after you start Planning Analytics Workspace Local.

See Backup or restore Planning Analytics Workspace Local for details on both the backup and restore procedures.

## What's new in books and reporting - 2.0.100

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.100.

### Discover AI-generated insights with Chart insights

A new feature, **Chart insights**, was made available in Planning Analytics Workspace 2.0.99 soon after its release. You can use **Chart insights** to get AI-generated insights in the form of summaries and key points to view an easier interpretation of a chart's data.

**Important:** The **Chart insights** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

For more information, see Get AI-generated insights with Chart insights.

## Perform AI-driven outlier analysis on explorations

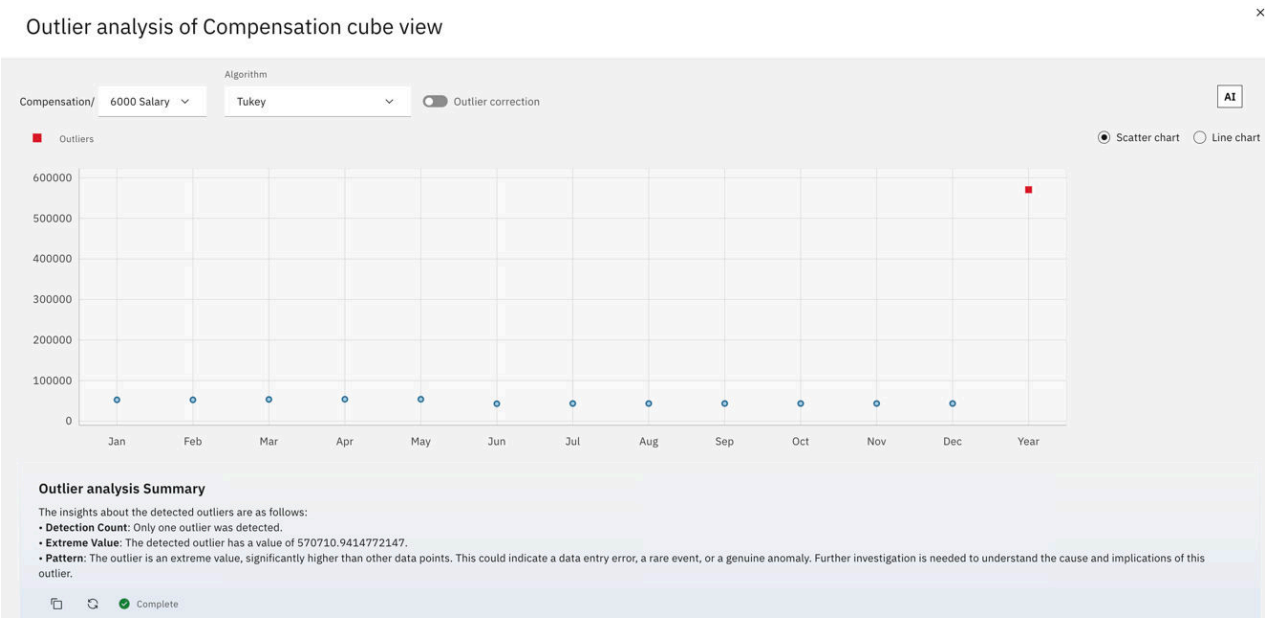
Use **Outlier analysis** to identify and correct data points in an exploration that deviate significantly from the norm, potentially indicating errors or anomalies. The analysis also includes AI-generated insights for any outliers that were detected and corrected in the view, helping you to investigate anomalies and improve the quality of your data.

The initial release of the **Outlier analysis** feature is available only in Planning Analytics Workspace Cloud and in English only.

**Important:** The **Outlier analysis** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

To perform an outlier analysis, click an exploration in a book and select the rows that you want to analyze. Then, click **AI operations** in the toolbar and select **Outlier analysis**.

You can select up to 25 rows to run **Outlier analysis** but the feature considers only the first 120 columns in the exploration for the analysis.




By default, the resulting analysis displays the data in the form of a Scatter chart, making it easy to see any outliers. The algorithm selected to conduct the outlier analysis is evaluated by using Silhouette score and the best ranked algorithm is then used to run the analysis. However, you can select a different algorithm and regenerate the analysis. You can also switch the chart type from a Scatter chart to a Line chart.

To correct the outliers, enable **Outlier correction**. The visualization adjusts by using a “normalized” data point for the outliers.

**Important:** Outlier correction does not modify the actual data point, that was identified as an outlier, on your Planning Analytics database.

An AI-generated summary of the outlier analysis automatically generates when you run the feature. If you have **Outlier correction** enabled, the summary includes insights for the outliers that were corrected.

For more information, see Outlier analysis.

## Use AI-driven impact analysis on explorations to identify key drivers

Use **Impact analysis** on an exploration to determine the impact of data fields on each other as well as identify and quantify relationships across the data. Generative AI interprets and summarized the analysis

to describe the direct and indirect impact of the data fields on each other, as well as any dependencies and relationships between them.

The initial release of **Impact analysis** is available only in Planning Analytics Workspace Cloud and in English only.

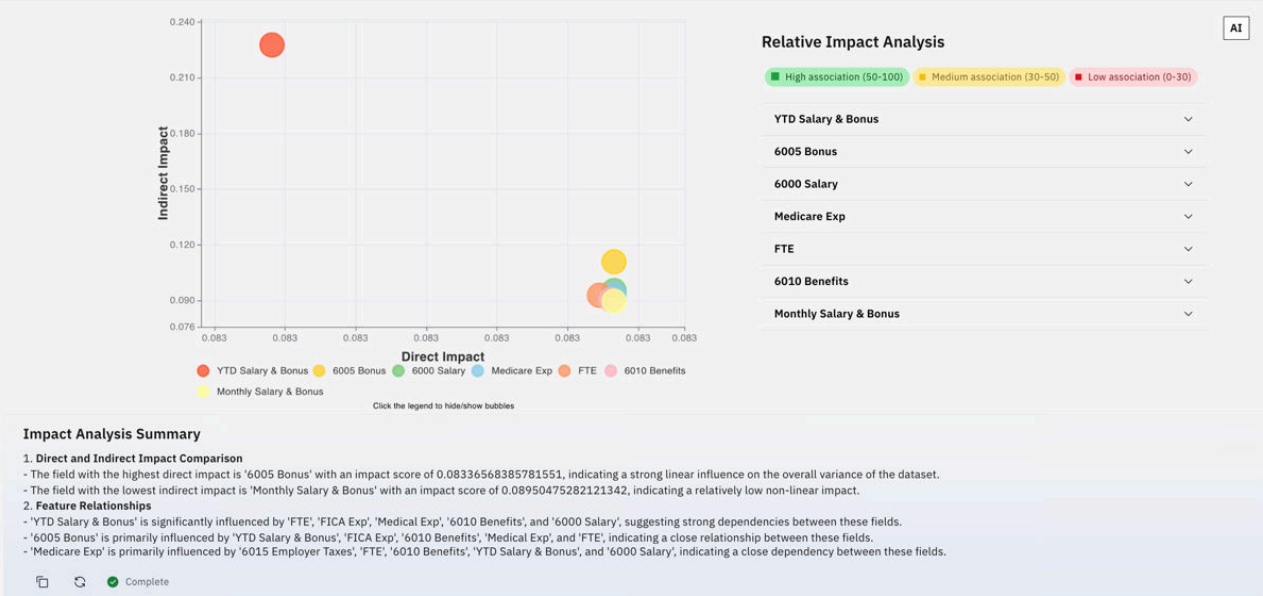
**Important:** The **Impact analysis** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

To start an **Impact analysis**, click an exploration in a book. Then, click **AI operations** in the toolbar and select **Impact analysis**.

To run the analysis, the exploration must have a minimum of two rows and six columns. The analysis is run on the first 120 rows and columns of the view.

The completed analysis results in a Bubble chart, which visually depicts the relationships between the data fields, along with an AI-generated summary of the analysis.

Impact Analysis of Compensation cube view




You can hover over a bubble to see the data field name and its direct and indirect impact values. You can also click the labels in the legend to show or hide the related bubbles in the chart.

When you click a bubble in the chart, the related section under **Relative impact analysis** expands to display the relative contribution of data fields on the selected data. The **Relative impact analysis** quantifies the degree of association between two data fields so that you can view the data that drives or influences the selected data.

For more information, see Impact analysis.

# What's new in plans and applications - 2.0.100

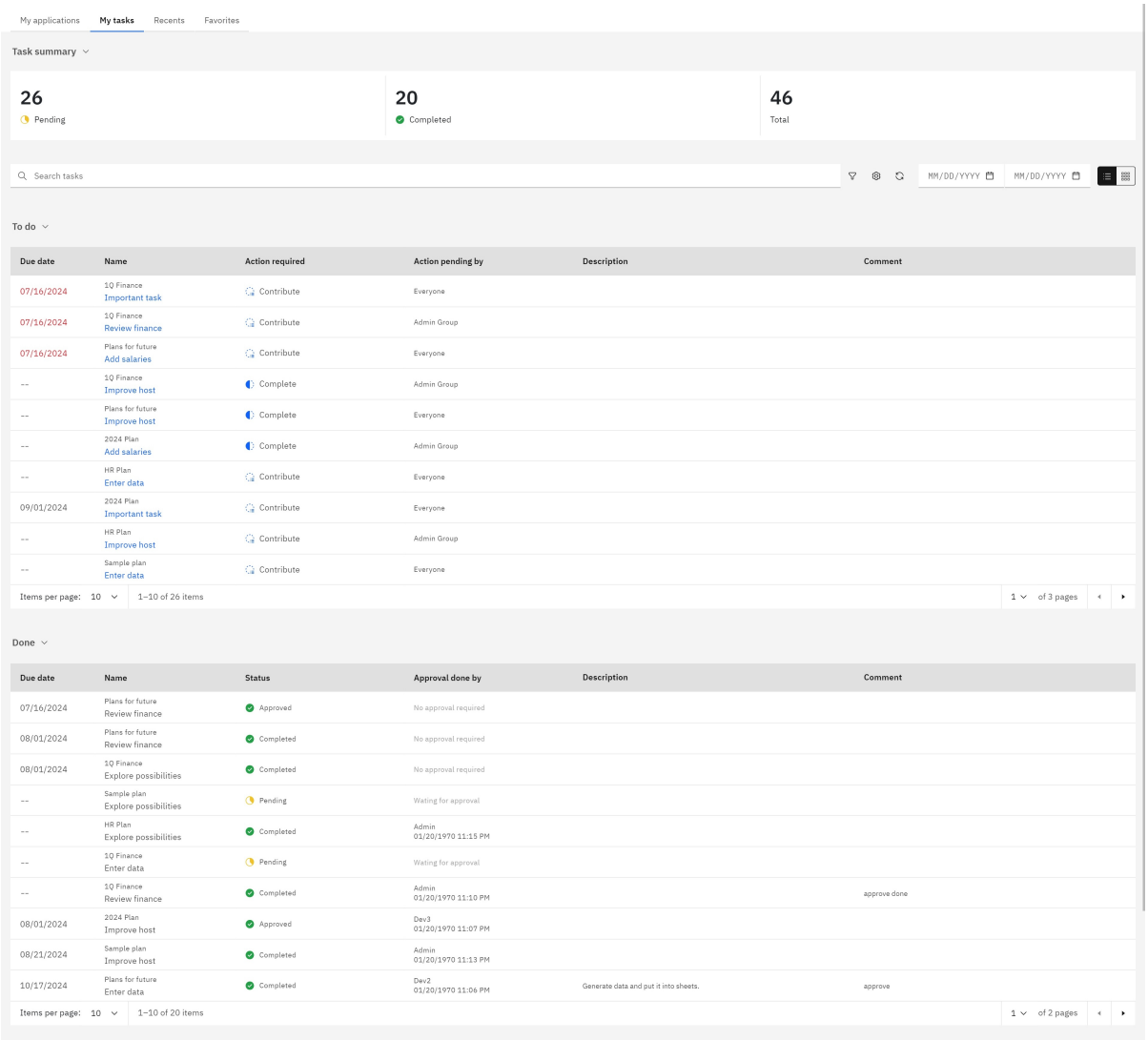
The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.100.

## My tasks tab available on the homepage

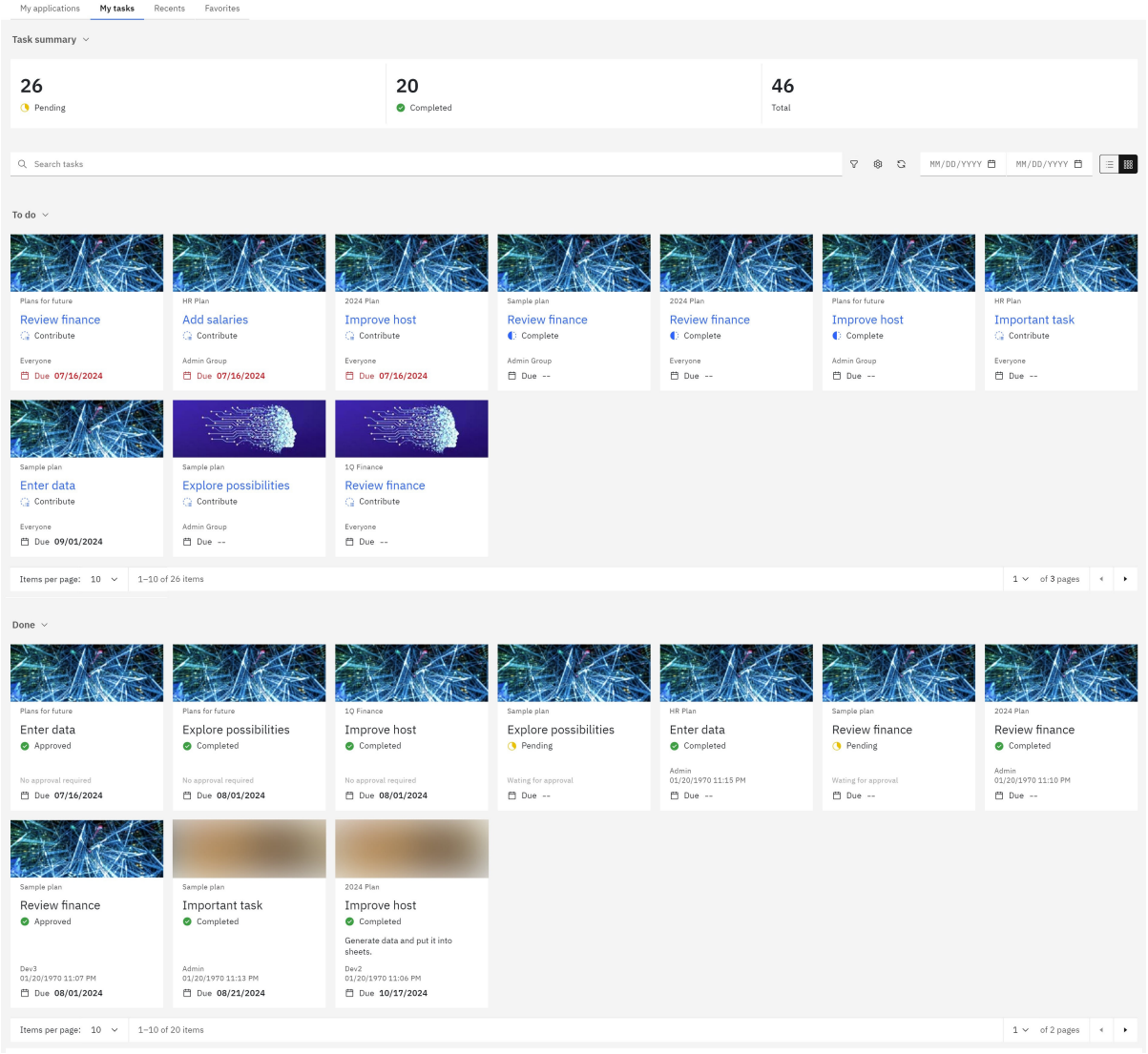
The Planning Analytics Workspace applications and plans homepage now has a tab called **My tasks**. The **My tasks** tab provides information about your current and completed tasks.

The **My tasks** tab is divided into three sections: **Task summary**, **To do**, and **Done**.

The following image is the **My tasks** tab on the Planning Analytics Workspace homepage in table view:




The following image is the **My tasks** tab on the Planning Analytics Workspace homepage in tile view:




## Task summary

The **Task summary** section provides at-a-glance information about your pending and completed tasks, allows you to search for tasks with certain criteria, and allows you to change the layout of the tab.

This section has the following features:

- The search bar allows you to search the content in the **To Do** and **Done** sections by task name.
- The filter icon allows you to filter the tasks in the **To Do** and **Done** sections by the action required and the status of the task.
- The date range selectors MM/DD/YYYY MM/DD/YYYY allow you to filter tasks by their due date. You can search for tasks due within a range of time, and the tab only displays tasks in that period.
- The view switcher allows you to change the layout between table view and tile view.
- The settings button has different options depending on the type of view you choose. If you choose the table view, you can adjust the size of the rows, and in tile view you can show or hide descriptions and thumbnails.
- The refresh button refreshes the page.

### To do

The **To do** section has information on tasks requiring further action.

The **To do** section has the following information:

- Due date: when the task is due.
- Name: the name of the task. Clicking on the name of a task opens it. From here, you can contribute, approve, or reject the task.
- Action required: the type of action required to advance the task to the next step.
- Action pending by: the users required to complete the action in the Action required section.
- Description: a description of the task or required action.
- Comment: comments added about the task or required action.

### Done

The **Done** section has information on the tasks you completed.

The **Done** section has the following information:

- Due date: when the task is due.
- Name: the name of the task.
- Status: the current status of the task.
- Approval done by: the user who approved the task. If the task has not been approved, this section will state that the task has not been approved, or if approval is not required.
- Description: a description of the task.
- Comment: comments added about the task.

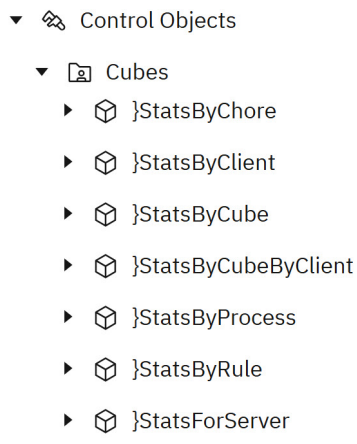
## What's new in modeling - 2.0.100

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.100.

### Manage performance monitoring from the modeling workbench (TM1 Database 11 only)

You can now start or stop performance monitoring for a database from the modeling workbench in Planning Analytics Workspace on TM1 Database 11.

When performance monitoring is enabled, performance and usage statistics are written to the `}Stats` control cubes for the database. You can use these control cubes to review minute-by-minute statistics for chores, clients, cubes, processes, rules, and the database (server). The `}Stats` cubes do not use much resources, their impact on overall memory usage is negligible.



To manage performance monitoring from the modeling workbench:

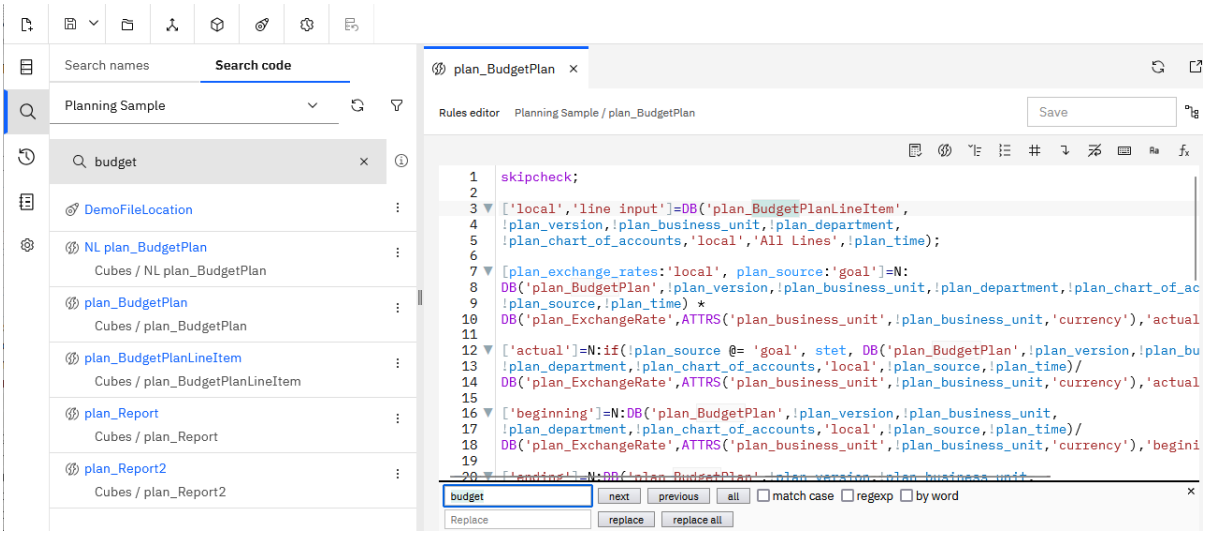
1. On the **Databases** tree, click the **Action** menu ⋮ for a database.
2. Click **Performance monitor**.
3. Enable or disable the **PerformanceMonitorOn** option.
4. Click **Save**.

The setting of the **PerformanceMonitorOn** option on the modeling workbench determines the status of the `PerformanceMonitorOn` configuration parameter for the database. Administrators can continue to manage the `PerformanceMonitorOn` configuration parameter in Planning Analytics Administration. The most recent setting for the `PerformanceMonitorOn` configuration parameter takes precedence, whether set on the modeling workbench or in Planning Analytics Administration.

## Search process and rule code

You can search process and rule code in the Planning Analytics database directly from the workbench.

The search panel in the workbench includes a new tab named **Search code**. On this tab, select a database and type your keywords. Use the filter next to the database name if you want to restrict the search to only processes or only rules. The default search includes both processes and rules.




The search results include the processes and rules that contain the specified keywords. When you click the result, the rule or process editor opens with the first instance of the keyword highlighted in the code. Use the buttons in the bottom panel of the editor to locate other instances of the keyword in the code.

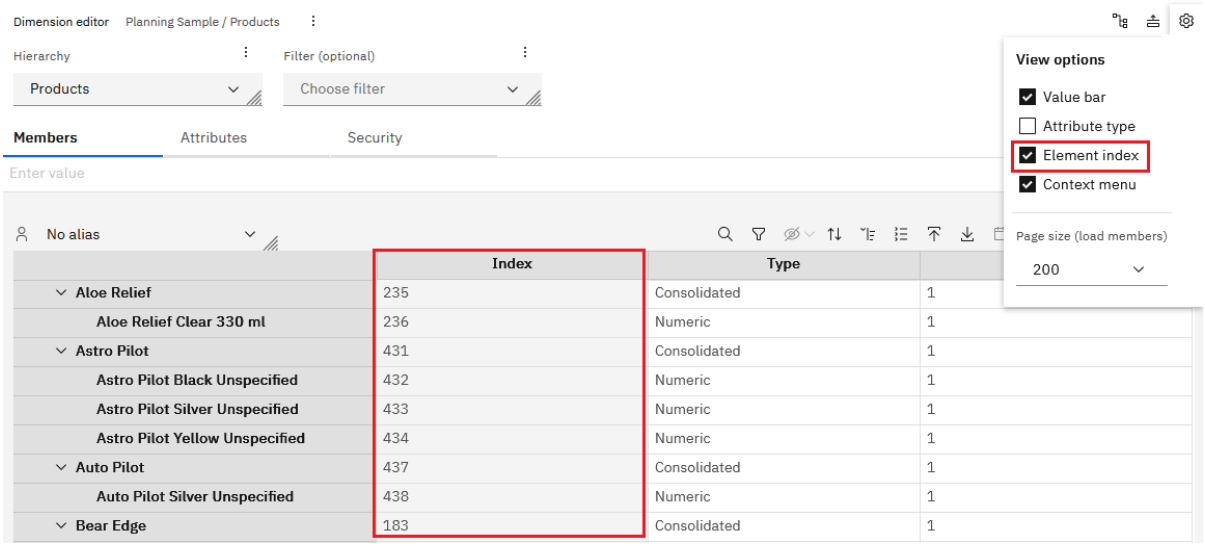
**Note:** The code for control processes is not searched in this release.

For more information, see Using database-level search in a modeling workbench.

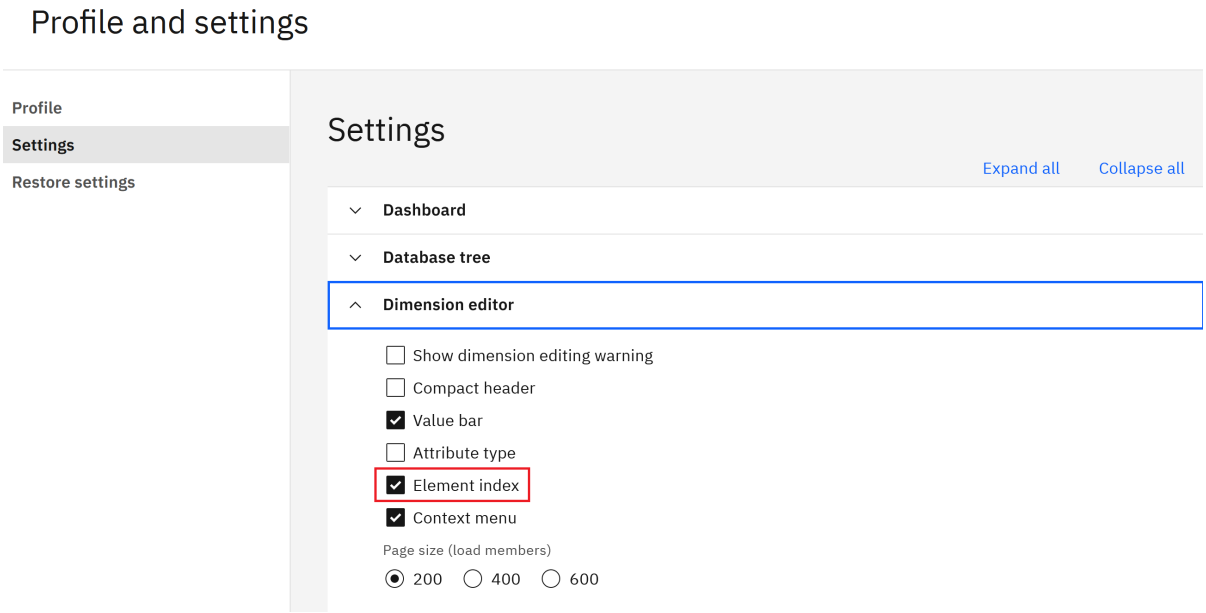
## Display the element index in the dimension editor

You can now display the element **Index** value in the dimension editor. **Index** is a property that determines the order of elements (members) within a dimension. The index value is unique for each element.

The **Settings** in the dimension editor now include the **Element index** option, which you can use to show or hide the index values for dimension elements (members). When you enable the **Element index** option, the dimension editor includes an **Index** column.




The **Element index** option is also available in the modeling workbench **Profile and settings**, under **Settings** > **Dimension editor**. Use this setting to enable this option by default for the entire workbench environment.




You can also sort the dimension elements by index with these new **Sort members** options:

**By index, ascending**
Sort members by index, in ascending order.

**By index, descending**
Sort members by index, in descending order.

**By index, hierarchically ascending**
Sort members within a hierarchy by index, in ascending order.

**By index, hierarchically descending**
Sort members within a hierarchy by index, in descending order.

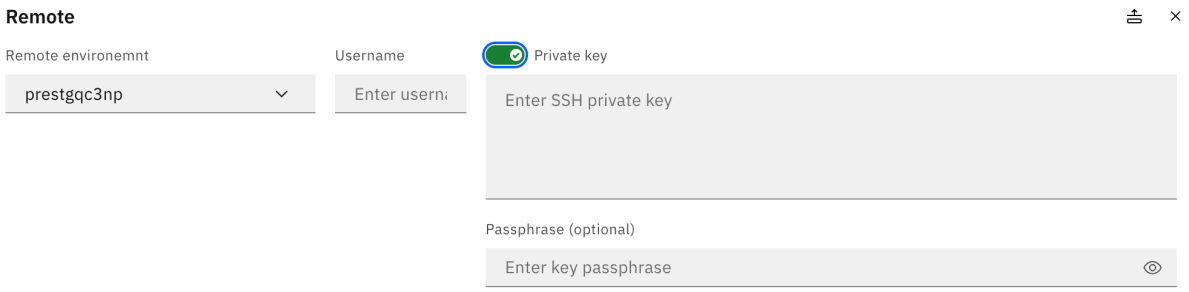
For more information, see Using the dimension editor.

## Drive explorer enhancements

New security and operation management features have been added to the **Drive explorer** in the IBM Planning Analytics Workspace modelling workbench.

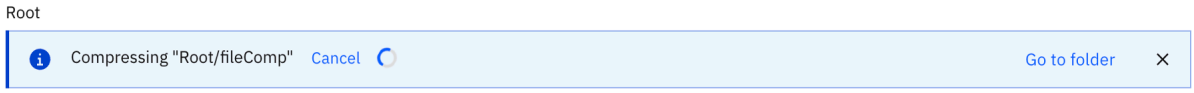
**Support for private keys and passphrases in Secure File Transfer Protocol (SFTP)**

You can now use private keys or passphrases in Secure File Transfer Protocol (SFTP). In the **Remote environment** tab in the drive explorer, click the **Private key** toggle. You can then enter a **Private key** or **Passphrase**.



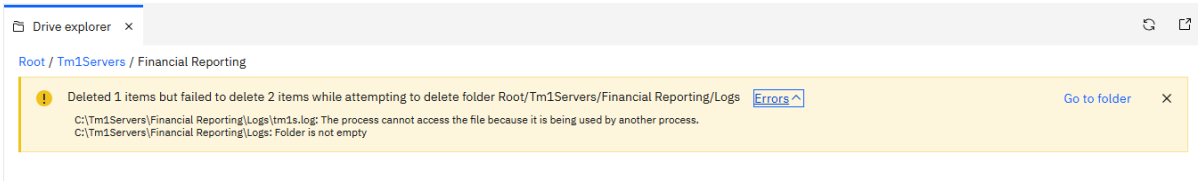

**Cancel current operation**

Using inline notifications, you can now cancel an operation while it is being performed. When an inline notification displays the status of an operation, click **Cancel** to stop the operation.




**Ignore locked files during copy, move, delete, and compress operations**

When you perform copy, move, delete, and compress operations, locked files do not prevent these operations from completing. Instead, after these operations are complete, a message lists the locked files that were not included in the operation.




For more information, see Access the S:/ drive and files in the remote server.

## Transaction log enhancements

Learn about the enhancements made to transaction logs in IBM Planning Analytics Workspace.

**Number of log entries displayed**

You can now increase the number of log entries on the screen by scrolling to the bottom of the list of entries.

The **Maximum entries** option has been removed, since there is no longer a maximum number of entries that can be displayed.

**Full query results with transaction downloads**

You can now download full query results. To download full query results, click the **Download** icon. Notifications appear when the download starts, and when it finishes.

**Filter based on Prior value and New value**

When performing a search, you can now filter by **Prior value** and **New value**.

**Note:** If the **Prior value** or **New value** you are inputting is a string value, start your query with `'`.

**Filter based on seconds**

When performing a search with a custom time period, you can now filter by seconds.

**Users selection enhanced**

When filtering a search by users, you can now find specific users using the new **Find users** option. To do this, click **Users/Chores** then click **Find users**. A list of users from the TM1 model is displayed. You can select one or more users to add to the filter.

**Select multiple log entries**

You can now select multiple log entries. To select multiple log entries, click on an entry, press the shift key, and select another entry. All entries in between are included in the selection.

# 2.0.99 - What's new, October 23, 2024

Learn about new features and known issues in version 2.0.99 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.99

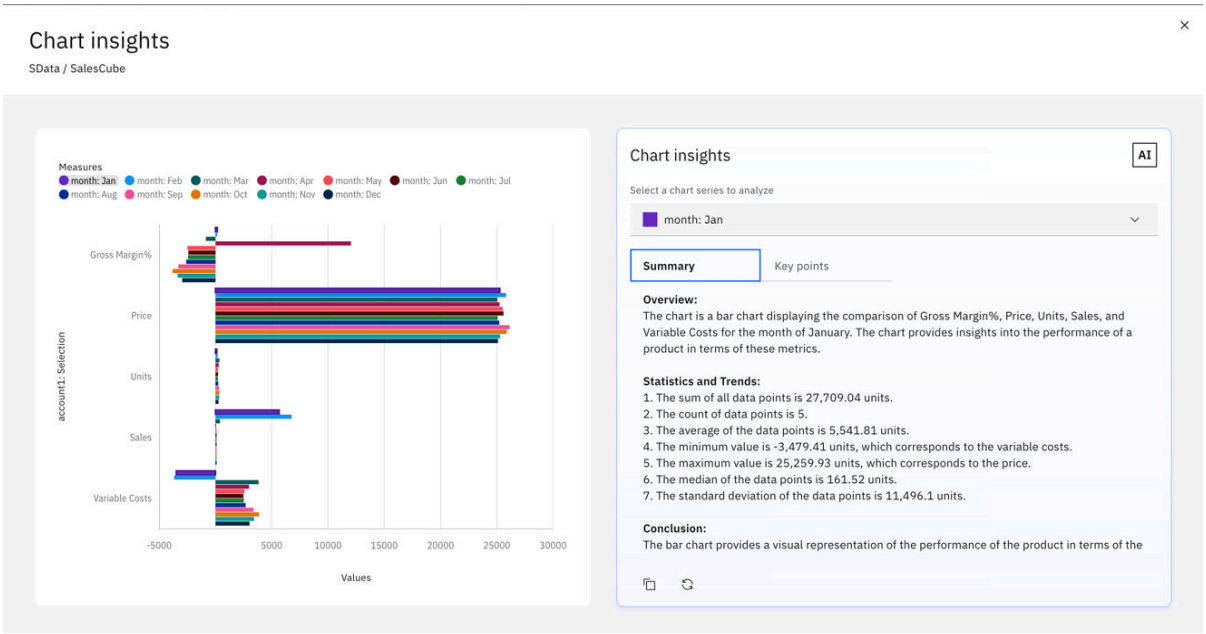
The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.99.

### Get AI-generated insights with Chart insights

A new feature, **Chart insights**, gives AI-generated insights in the form of summaries and key points to provide an easier interpretation of the chart data. The initial release of the **Chart insights** feature supports English only and is available only in Planning Analytics Workspace Cloud.

**Important:** The **Chart insights** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

With **Chart insights**, depending on the chart type and the chart data, you can get an overview of the chart, statistics and trends, trend analysis, insights, notable observations, and more. The **Summary** tab provides a more detailed analysis of the chart than the **Key points**.

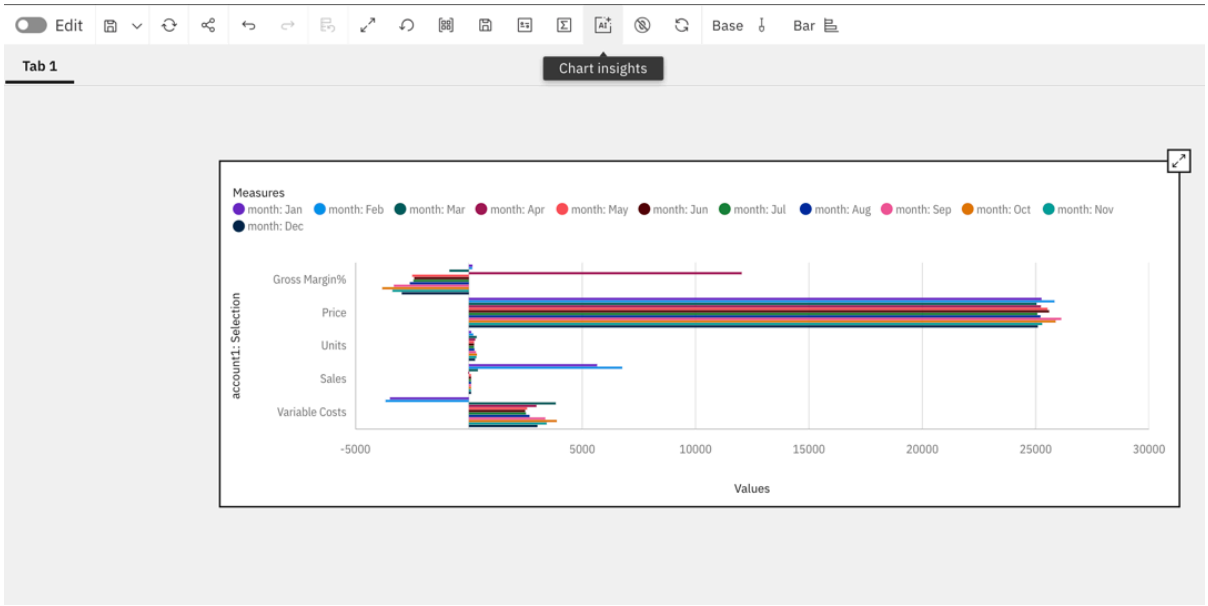



You can view details of the AI model and how it works in this feature by clicking the **AI** label.

## Generating insights

To generate AI insights for a chart:

1. Select the chart that you want the analysis for and click **Chart insights** in the toolbar.




2. If there are multiple chart series in your chart, select a chart series to generate the AI insights.

AI starts generating a text summary of the data in the selected chart series. After the summary is complete, you can go to the **Key points** tab for insights and notable observations.

**Note:** You cannot navigate between the **Summary** and **Key points** tabs during text generation. You also cannot regenerate or copy the response until after generation is complete.

**Chart insights** supports the following charts for analysis:

- Area
- Bar

- Box plot
- Dual axes column
- Dual axes line
- Line
- Line and column
- Pie
- Point
- Radar
- Radial
- Stacked bar
- Stacked column
- Waterfall
- Word cloud

### Best practices for generating insights

Here are a few things that you can do that might lead the AI model to generate better insights:

- Chart has data and is not empty
- Chart data is in context of the cube
- Cube, dimension, and member names are descriptive and accurately represent the data
- There are no named outliers for dimensions that might confuse the AI model

For more information, see Use AI-generated insights to interpret a visualization.

## Extended language support for Planning Analytics AI assistant

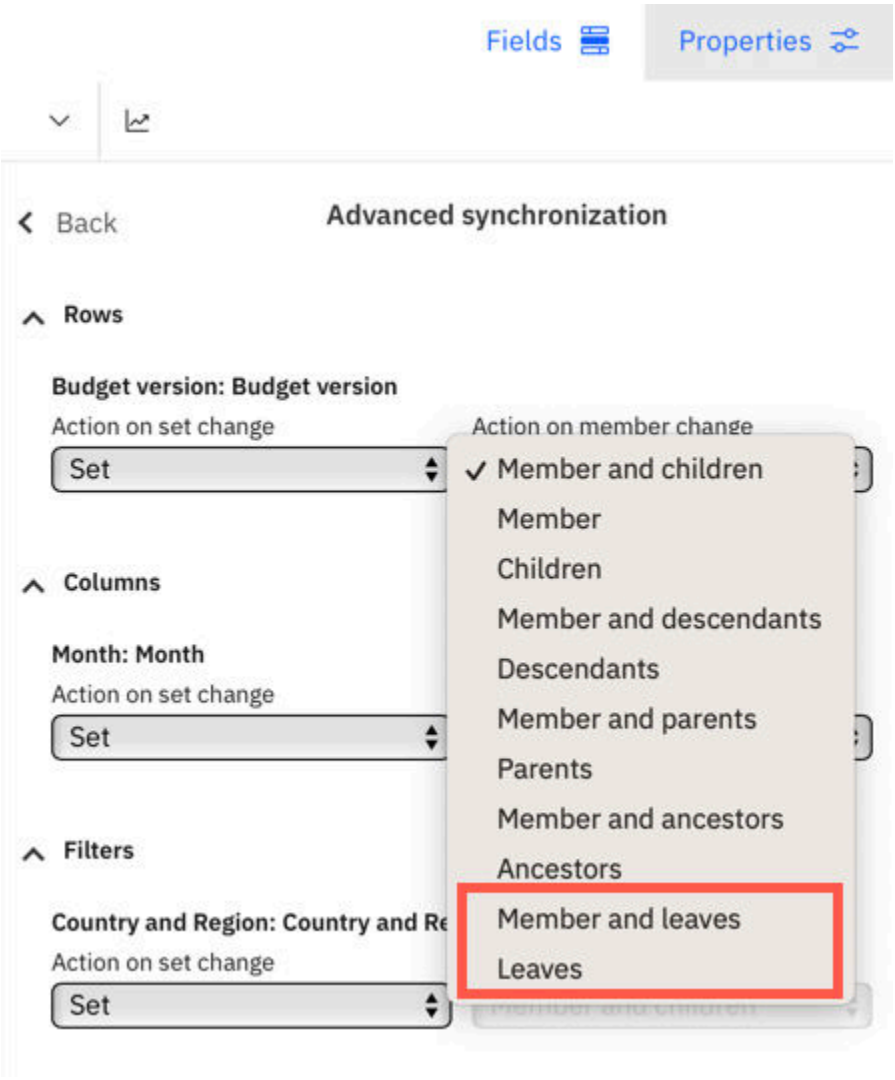
The Planning Analytics AI assistant now supports English, Spanish, French, and German.

If you're using Planning Analytics Workspace in any other language, the Planning Analytics AI assistant defaults to English.

## New advanced synchronization options for member change

Two new options, **Member and leaves** and **Leaves**, are now available for member change events under **Advanced synchronization**.

The **Advanced synchronization** feature allows you to customize how each hierarchy in the cube viewer or visualization displays when a synchronization event is received by that widget. When a member change synchronization occurs, in addition to other options, you can now choose to display the member and its leaves or just the leaves in the widget that is receiving the synchronization event.



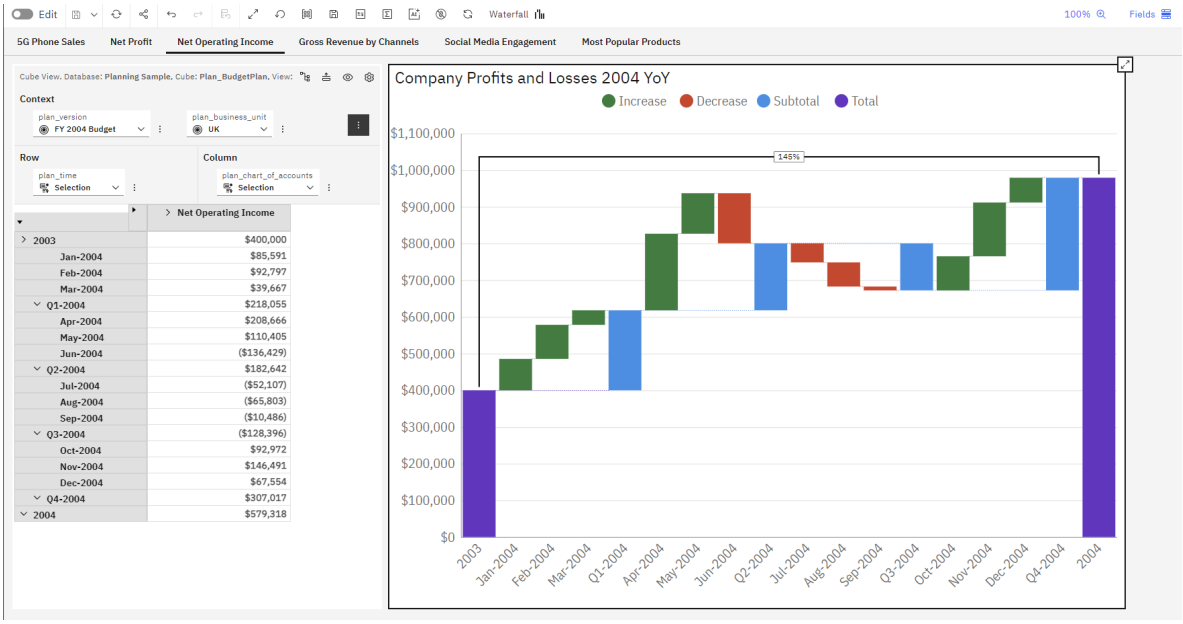

For more information, see Setting synchronization for objects.

## New waterfall visualization

New type of waterfall visualization is now available.

The new visualization is named **Waterfall**. The preexisting waterfall visualization is renamed to **Legacy waterfall**, and can still be used.

The following example shows a typical waterfall visualization. This visualization is used to show some kind of a measure (`Net Operating Income` in the example) over a period of a year. The previous year (2003) is shown as a starting point. Then the visualization lists the numbers for each month or quarter, and ends with the total for the entire year (2004).




When comparing the new waterfall visualization to the legacy waterfall visualization, the following two options in the new visualization significantly improve the user experience:

- The **Change the bar type as** option in the bar right-click menu.

  Use this option to change the bar type to one of the following types: **Auto**, **Delta**, **Adjustment**, or **Subtotal**.

- The **Accumulate totals** property under **Visualizations** > **Data**.

  Use this property to correctly render values from multiple **Total** bars in the visualization instead of changing the TM1 data.

No upgrade process is provided for replacing the legacy waterfall visualization with the new waterfall visualization in existing content. If you want to use the new visualization in existing views, you need to reopen the book and manually replace the legacy waterfall with the new waterfall visualization.

For more information, see Waterfall.

## Use action buttons to synchronize groups and hierarchies on navigation

With the new **Advanced context** feature in action buttons, you can now pass context and synchronize groups between books on navigation. Widgets in the same group synchronize together. You can also select which hierarchies are used to pass context.

Previously, action buttons depended on the default synchronization group to provide context to the same synchronization group in the target book. These groups had to have names that were an exact match and the context of all common hierarchies from the default synchronization group were passed to the target.

With **Advanced context**, you can map a source synchronization group to a target synchronization group and select which hierarchies that are common to both groups, receive context. You can configure multiple action buttons to pass context to a single target book and drill through data easily.

Before you can map a synchronization group from the source book to the synchronization group in the target book, you need to first set up synchronization and the widgets in both books need to be of the same synchronization group. **Advanced context** supports all three synchronization groups (Book, Sheet, and Custom).

To map synchronization groups and configure context passing, follow these steps:

1. In edit mode, add an action button to the source book.
2. Go to the action button's **Properties** and under **Navigation**, enable **Navigate to destination**.
3. Click **Other book** and select a target book and the tab that you want to pass context to.

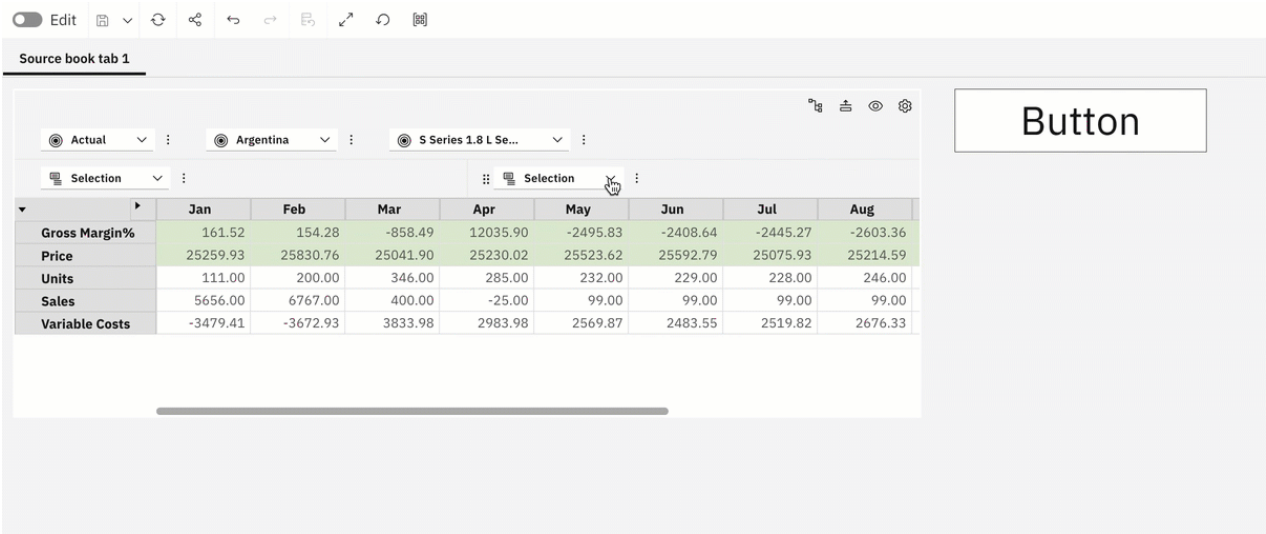
4. Click **Pass context**.

   The **Advanced context** link becomes available.
5. Click **Advanced context** and select **Custom** group mapping.

   **Note:** The **Default** group mapping displays the default synchronization groups and hierarchies. To edit or configure these settings, go to **Custom** group mapping.
6. Select the source group, which passes the context, the target group that receives the context, and the hierarchies you want to synchronize.

   You can add more mappings or remove existing ones under **Custom**.

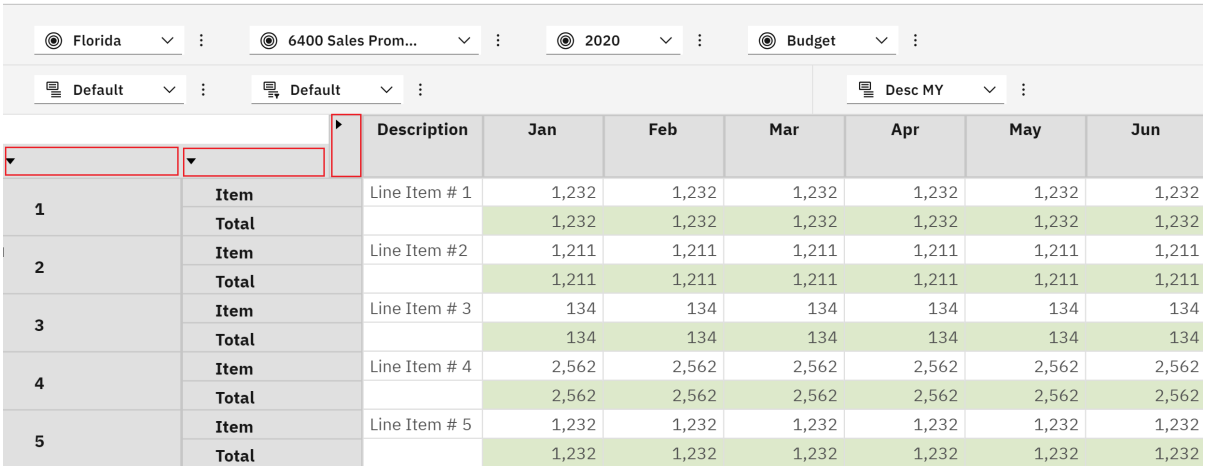
Now, change the hierarchies in a widget in the source book and click the action button. You automatically navigate to the target book and hierarchies that were selected to synchronize, update in the widget.




## Thumb cells formatting options for a view

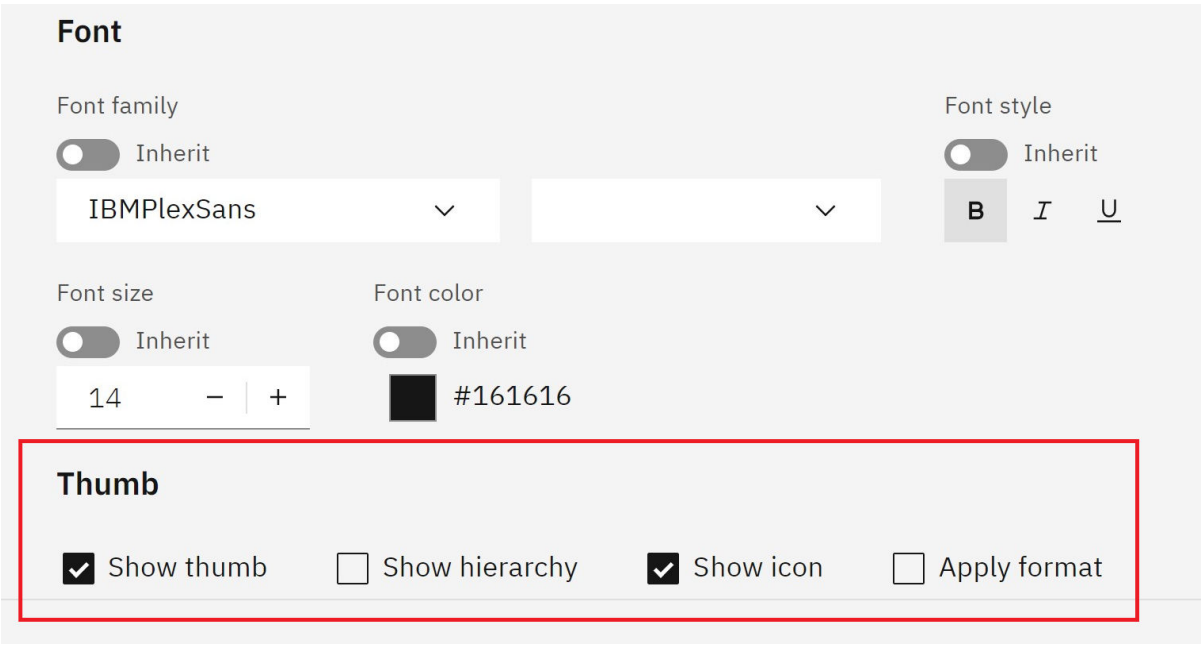
Format manager now includes options to format the thumb cells in a view.

The thumb cells are located on the row and column axis of the cube viewer, on top of the row headers and to the left of the column headers, as highlighted in the following image:




Until now, these cells could not be formatted. For example, no text could be entered as a column header, the cell background was always gray, and the arrow icons were always present. Starting with this release, users can change the display of these cells by using the **Thumb** formatting options in **Format manager**, in the **Column header** and **Row header** sections.

The following image shows the **Thumb** options in the Format manager. These options are identical for column headers and row headers.




Use these options to apply the following changes to the thumb cells:

**Show thumb**
Use this option to hide the thumb cells on top of the row headers and to the left of the column header. You can enable the thumb cells for just the row header, column header, or both. This option is enabled by default.

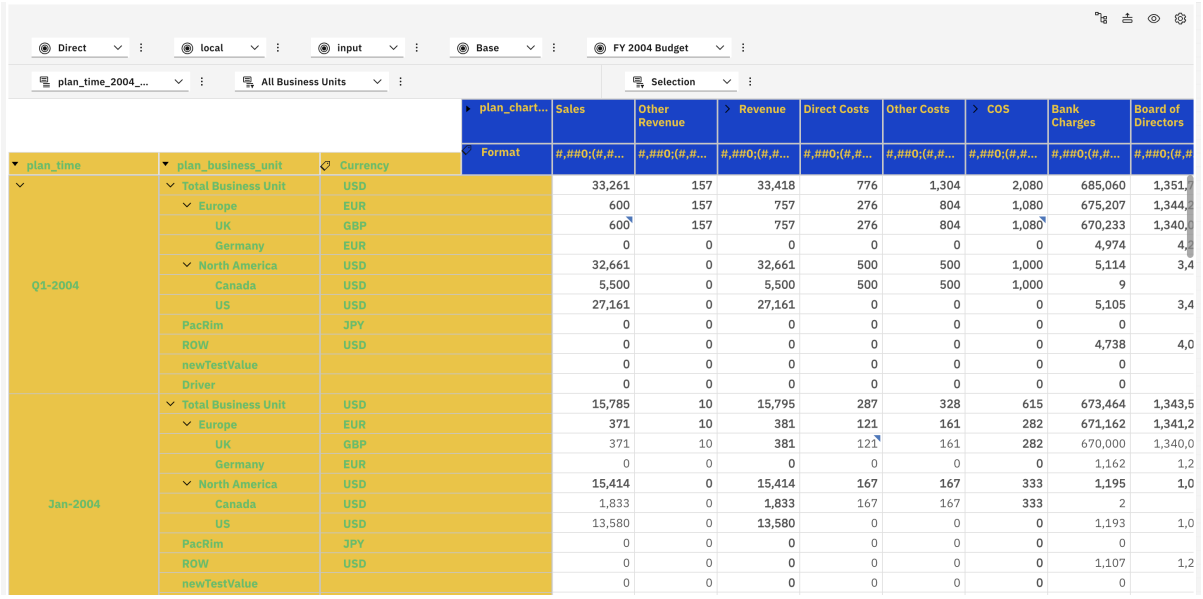
**Show hierarchy**
Use this option to show the hierarchy name per column axis or row axis. You might want to show the hierarchy name in the row header thumb cells, but not the column header thumb cells (you can leave the column header thumb cells displayed without the hierarchy name).

**Note:** This option controls the display of the attribute headings, if you used the **Show attributes** option with the thumb cells. These headings are displayed only when **Show hierarchy** is selected.

**Show icon**
Use this options to show or hide the arrow icon in the thumb cells. This option is enabled by default.

**Apply format**
Use this option if you want to apply the column header format to the column thumb cells, and the row header format to the row thumb cells.

The following image includes an example of thumb cells formatting:



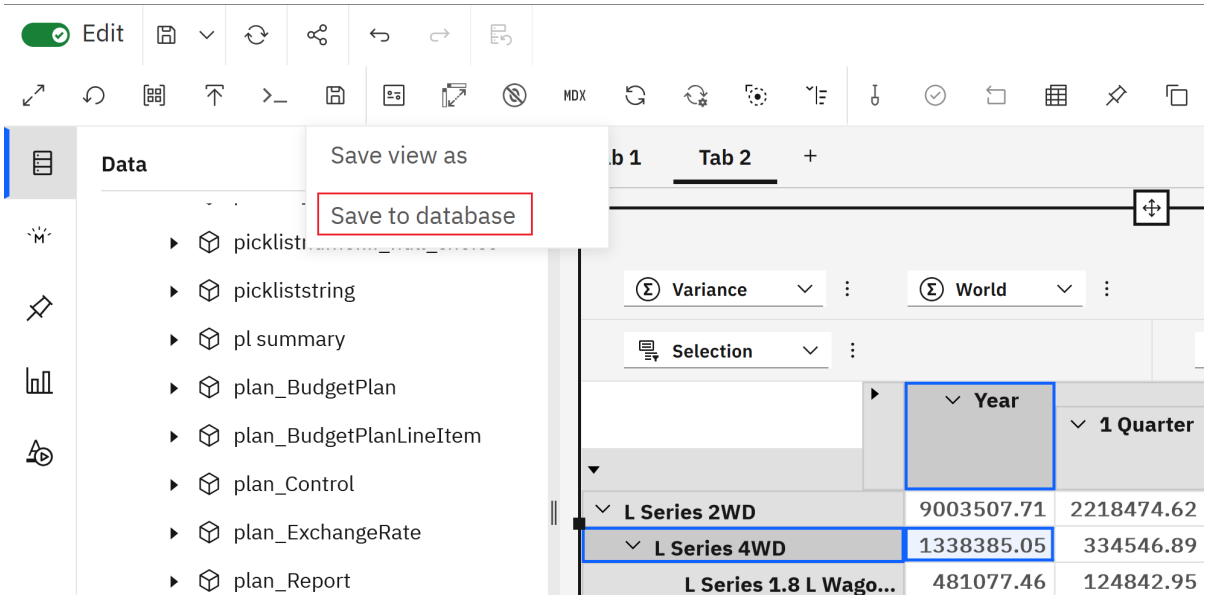

| plan_time | plan_business_unit | Currency | Sales | Other Revenue | Revenue | Direct Costs | Other Costs | COS | Bank Charges | Board of Directors |
|---|---|---|---|---|---|---|---|---|---|---|
| | | Format | #,##0;(#,#... | #,##0;(#,#... | #,##0;(#,#... | #,##0;(#,#... | #,##0;(#,#... | #,##0;(#,#... | #,##0;(#,#... | #,##0;(#,# |
| Q1-2004 | Total Business Unit | USD | 33,261 | 157 | 33,418 | 776 | 1,304 | 2,080 | 685,060 | 1,351, |
| | Europe | EUR | 600 | 157 | 757 | 276 | 804 | 1,080 | 675,207 | 1,344, |
| | UK | GBP | 600 | 157 | 757 | 276 | 804 | 1,080 | 670,233 | 1,340, |
| | Germany | EUR | 0 | 0 | 0 | 0 | 0 | 0 | 4,974 | 4, |
| | North America | USD | 32,661 | 0 | 32,661 | 500 | 500 | 1,000 | 5,114 | 3,4 |
| | Canada | USD | 5,500 | 0 | 5,500 | 500 | 500 | 1,000 | 9 | |
| | US | USD | 27,161 | 0 | 27,161 | 0 | 0 | 0 | 5,105 | 3,4 |
| | PacRim | JPY | 0 | 0 | 0 | 0 | 0 | 0 | 0 | |
| | ROW | USD | 0 | 0 | 0 | 0 | 0 | 0 | 4,738 | 4,0 |
| | newTestValue | | 0 | 0 | 0 | 0 | 0 | 0 | 0 | |
| | Driver | | 0 | 0 | 0 | 0 | 0 | 0 | 0 | |
| Jan-2004 | Total Business Unit | USD | 15,785 | 10 | 15,795 | 287 | 328 | 615 | 673,464 | 1,343,5 |
| | Europe | EUR | 371 | 10 | 381 | 121 | 161 | 282 | 671,162 | 1,341,2 |
| | UK | GBP | 371 | 10 | 381 | 121 | 161 | 282 | 670,000 | 1,340,0 |
| | Germany | EUR | 0 | 0 | 0 | 0 | 0 | 0 | 1,162 | 1,2 |
| | North America | USD | 15,414 | 0 | 15,414 | 167 | 167 | 333 | 1,195 | 1,0 |
| | Canada | USD | 1,833 | 0 | 1,833 | 167 | 167 | 333 | 2 | |
| | US | USD | 13,580 | 0 | 13,580 | 0 | 0 | 0 | 1,193 | 1,0 |
| | PacRim | JPY | 0 | 0 | 0 | 0 | 0 | 0 | 0 | |
| | ROW | USD | 0 | 0 | 0 | 0 | 0 | 0 | 1,107 | 1,2 |
| | newTestValue | | 0 | 0 | 0 | 0 | 0 | 0 | 0 | |

For more information, see Format cube views with the Format manager.

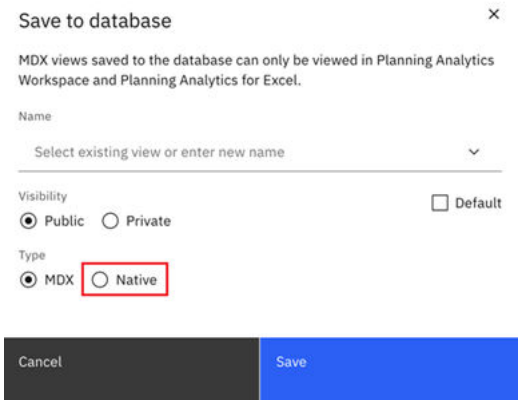
## Saving a cube view as a native view

You can now save a cube view as a native view.

The **Save to server** menu option and dialog box was renamed to **Save to database**.




The dialog box now includes the **Native** option which allows you to save an MDX view as a native TM1 view.




**Note:** When a view contains a calculation, it cannot be saved as a native view.

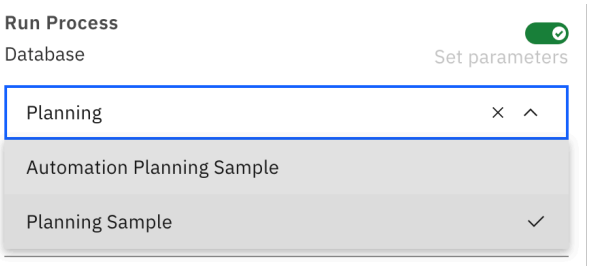
In the **Name** field, you can rename the view, or select another view from the drop-down menu, which shows names of other existing views. If you select the **Default** checkbox, the **Name** input field is disabled, and the current view is selected as the default view.

For more information, see Save a view.

## Action button search function

The search function for selecting databases and processes for an Action button in IBM Planning Analytics Workspace has been improved.

**Search function filters**
When you type the name of a process or database, the search function now filters for results with the same name.




**Regular processes and control processes now included in search results**
When you search for processes, both regular TI processes and control processes are shown in the results.

For more information on action buttons, see Action buttons.

## Changed Selector menu options

The menu options for adding the **Selector** widget are changed. These options are now available also from the **Data** tree.

The options are consistent in the metadata tree, the cube viewer, and the **Fields** panel.

The previous options **Add as selector widget** (metadata tree) and **Add widget as** (cube viewer and **Fields** panel) are replaced by the following two options:

- **Add selector tile**

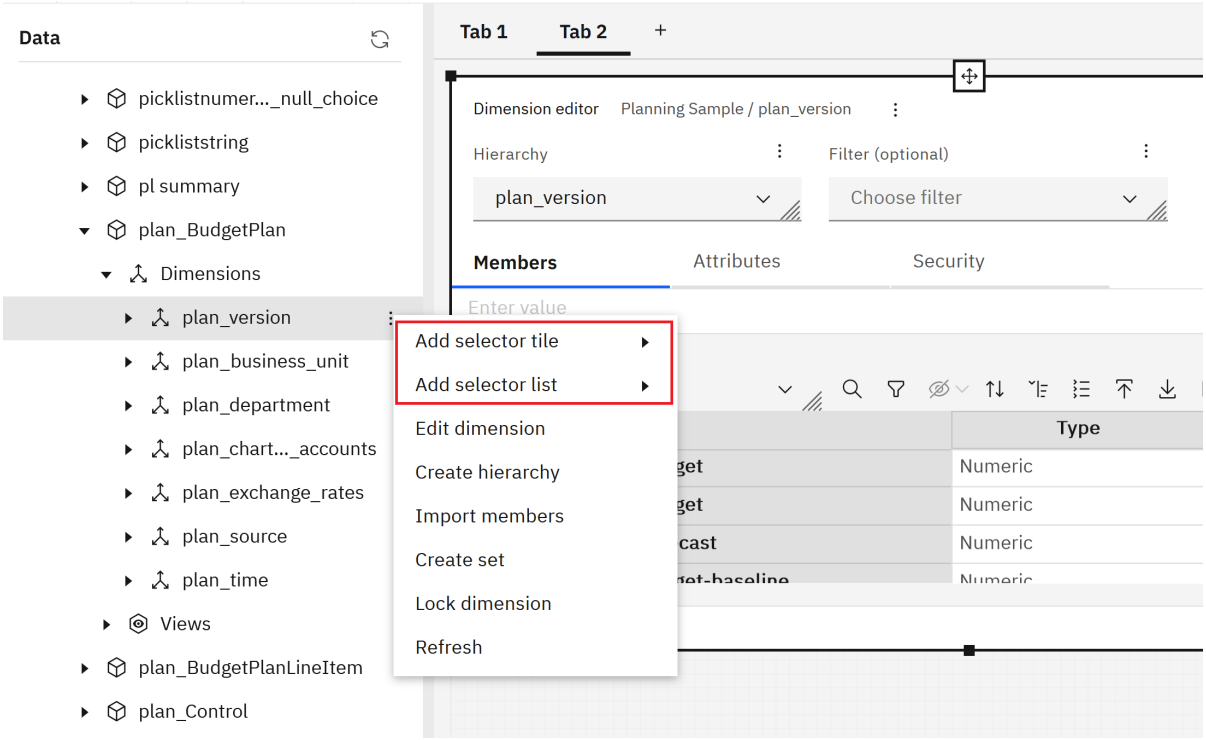
  Use to add the selector widget as a tile. You can add 3 types of tiles: **Single tile**, **Multiple tiles**, and **Range selection tile**.
- **Add selector list**

Use to add the selector widget as a list. You can add 3 types of lists: **Single list**, **Multiple lists**, and **Multiple selection list**.

The new menu options are available from the following context menus in the **Data** tree, the cube viewer, and the **Fields** panel:

- Dimension context menu
- Hierarchy context menu
- Sets context menu

In the following example, the new menu options are shown from the dimension context menu in the data tree:




For more information, see Create a Selector visualization to manage views.

## Views exported to Excel indented based on member relationship

All cube views that you export to Excel now include indentation on rows where there is a parent-child relationship between the members even if the rows aren't indented in Planning Analytics Workspace.

For information about exporting a view from Planning Analytics Workspace to Excel, see Export to Excel.

# What's new in modeling - 2.0.99

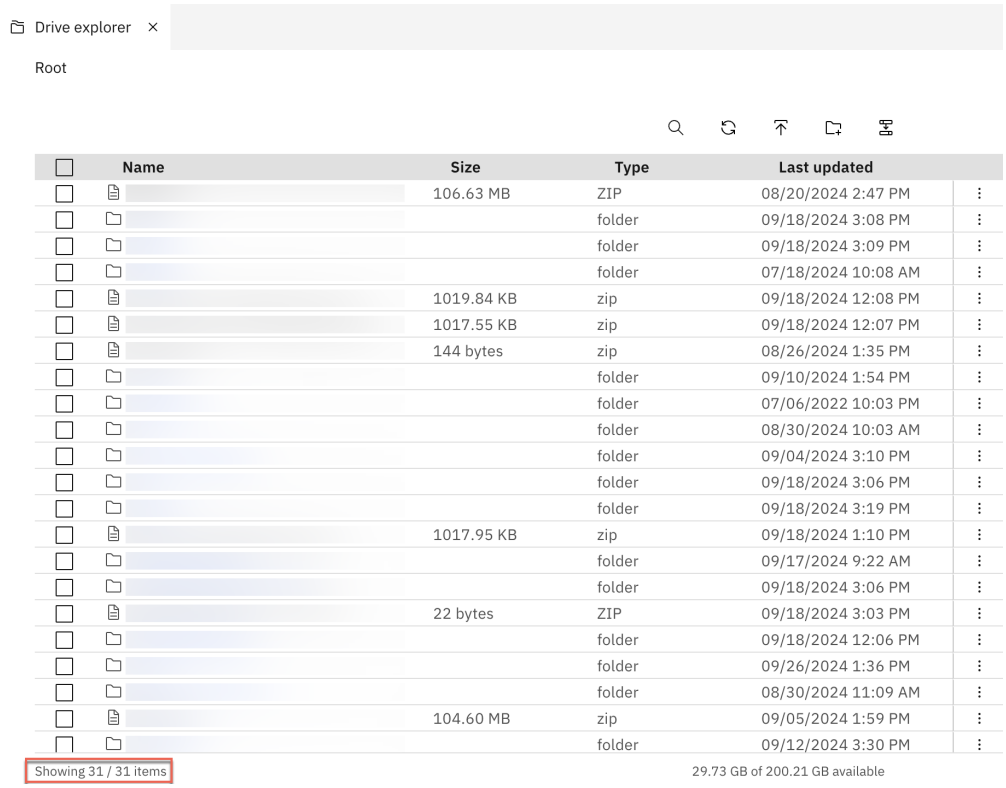
The following new and changed modeling features are available in Planning Analytics Workspace 2.0.99.

## Drive explorer enhancements

New features have been added to the **Drive explorer** in the IBM Planning Analytics Workspace modelling workbench.

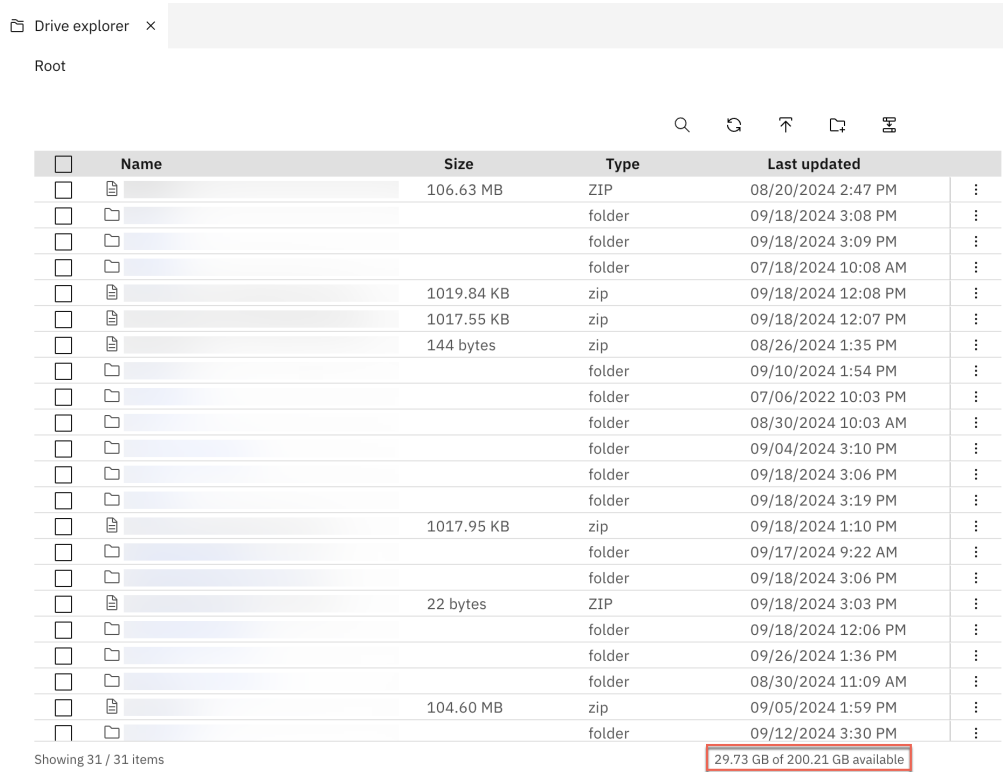
**View the number of files in a directory**

When you are in a directory, you can now view the number of items in that directory. This feature is helpful for providing at-a-glance information about the files in large directories. For example, you can see the number of log files you have in a logging directory.



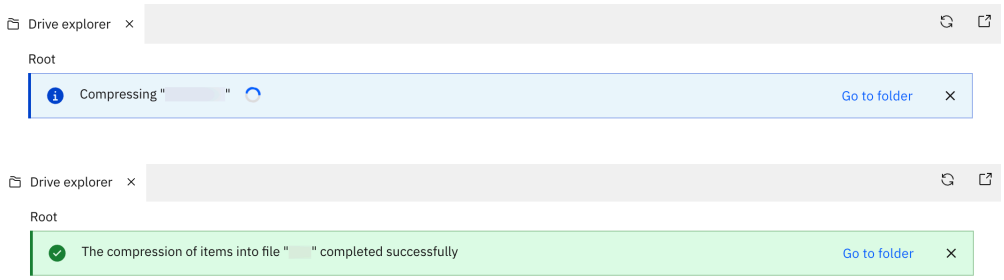

## View disk space

You can now view the total amount of disk space you have and the amount of disk space you have left.




## Inline notifications

The drive explorer now displays inline notifications about the status of actions you perform. For example, when compressing files, an inline notification appears when the process is being performed, and when it is complete.




**Unzipping files**

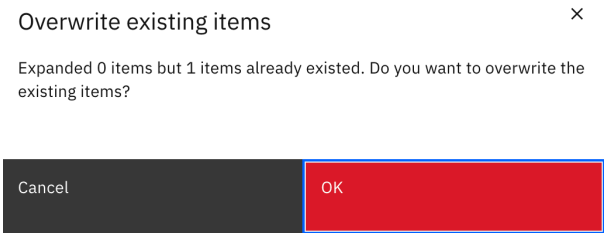
The process for unzipping files has improved. You can now expand a file into the directory you are viewing, or into a folder.

To expand a file into the directory you are viewing, click **Expand**.

To expand a file into a folder, click **Expand to**, name the folder, and click **Create**. The contents of the unzipped file are in the newly created folder.

**Overwrite notification when expanding a file**

If you expand a file to an existing folder with the same name, the following notification appears:




You cannot choose which files are overwritten. The notification appears only once.

**Tm1s.log**

The `tm1s.log` file is now visible in the directory. The file's presence can be seen, but it cannot be opened or deleted.

**Improved performance**

The performance of drive explorer is improved when displaying a large number of files and folders.

## Improved verification of valid file types on file upload

On file upload, IBM Planning Analytics Workspace now determines the validity of a file based on the file type instead of the file extension.

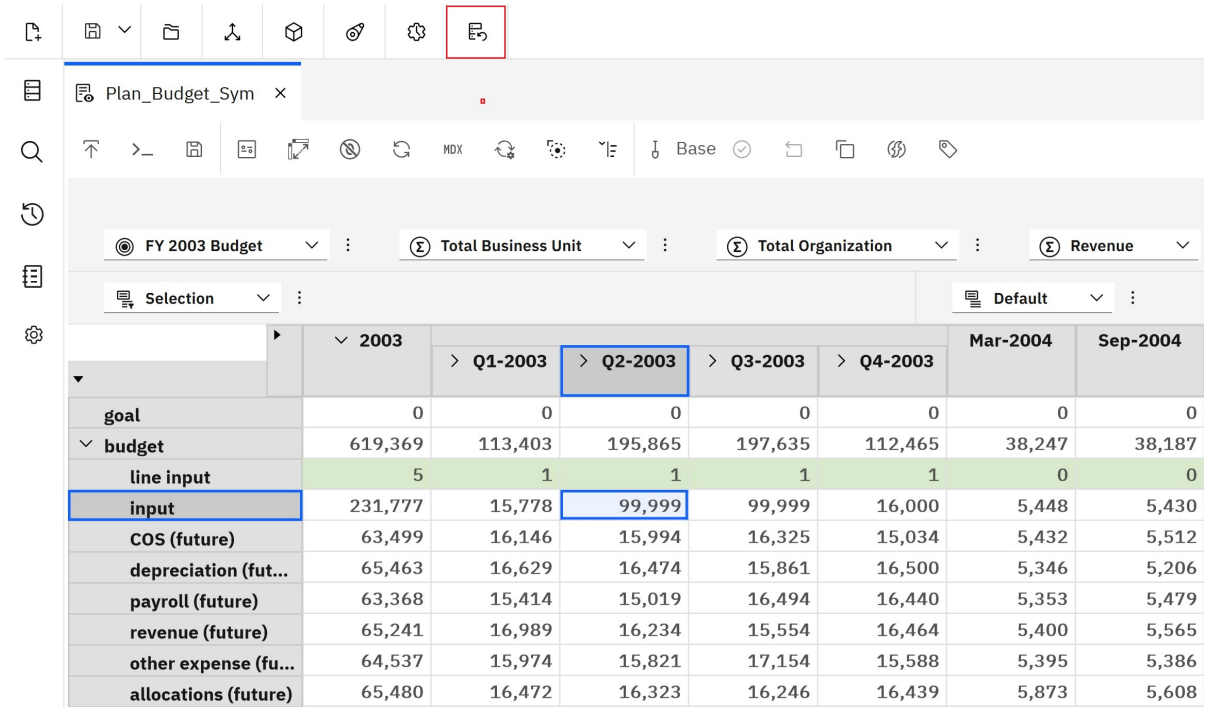
Some of the file types that are supported include:

- Text files (such as .txt, .csv, .css, .html)
- Application files (such as .docx, .json, .ics, .xml, .php, .pdf, .zip, .7z, .rtf, .rar, .xlsm)
- Image files (such as .jpeg, .jpg, .svg, .gif, .png, .heic)
- Font files (such as .tff,. woff, .otf)
- Video files (such as .mp4, .mpeg, .avi)
- Audio files (such as .mp3, .wav, .weba, .mid)

## Roll back value changes in a view

Use the **Roll back data entry** button to roll back value changes in a view.

The **Roll back data entry** button is located on the workbench toolbar.




| | 2003 | Q1-2003 | Q2-2003 | Q3-2003 | Q4-2003 | Mar-2004 | Sep-2004 |
|---|---|---|---|---|---|---|---|
| goal | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| budget | 619,369 | 113,403 | 195,865 | 197,635 | 112,465 | 38,247 | 38,187 |
| line input | 5 | 1 | 1 | 1 | 1 | 0 | 0 |
| input | 231,777 | 15,778 | 99,999 | 99,999 | 16,000 | 5,448 | 5,430 |
| COS (future) | 63,499 | 16,146 | 15,994 | 16,325 | 15,034 | 5,432 | 5,512 |
| depreciation (fut... | 65,463 | 16,629 | 16,474 | 15,861 | 16,500 | 5,346 | 5,206 |
| payroll (future) | 63,368 | 15,414 | 15,019 | 16,494 | 16,440 | 5,353 | 5,479 |
| revenue (future) | 65,241 | 16,989 | 16,234 | 15,554 | 16,464 | 5,400 | 5,565 |
| other expense (fu... | 64,537 | 15,974 | 15,821 | 17,154 | 15,588 | 5,395 | 5,386 |
| allocations (future) | 65,480 | 16,472 | 16,323 | 16,246 | 16,439 | 5,873 | 5,608 |

This button is disabled when none of the views in the workbench contains changed values. As soon as a value is changed in any view, the button is enabled and remains enabled as long as any view contains changed values. Even if you save and close the view with the changed value, and save and close the workbench, the button remains enabled unless you commit the data.

The button rolls back values starting with the most recent change in any view, no matter if the view is open or closed, including when the view is open in a separate window, or the value is in a single cell visualization. You need to click the button to roll back each change. You cannot roll back all changes in one click.

For consolidated values, all dependent values are also rolled back.

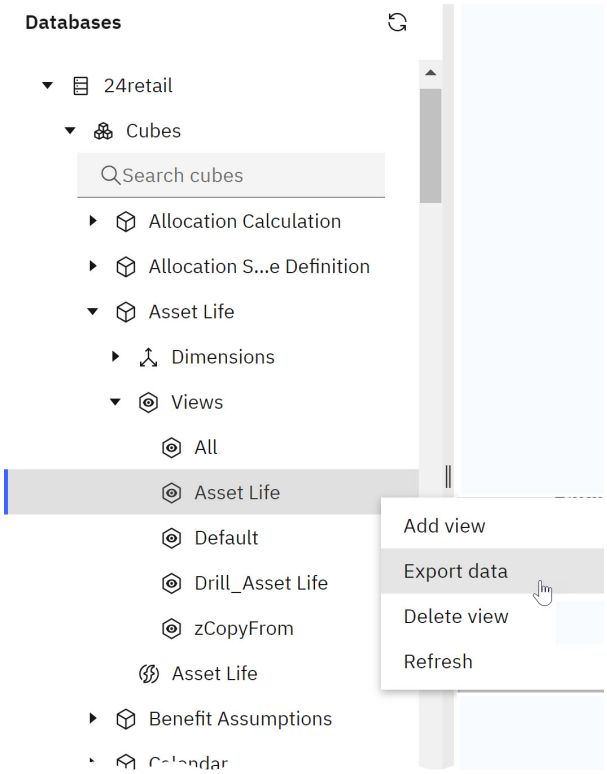
For more information, see Roll back data entry.

## Export data from a cube or view on a modeling workbench

You can now export data from a cube or view to a `.csv` file. The exported file is saved to your local `Downloads` directory.

To export data from a cube, right-click the cube on the modeling workbench **Databases** tree, then click **Export data**.

To export data from a view, right-click the view on the modeling workbench **Databases** tree, then click **Import/Export** > **Export data**.



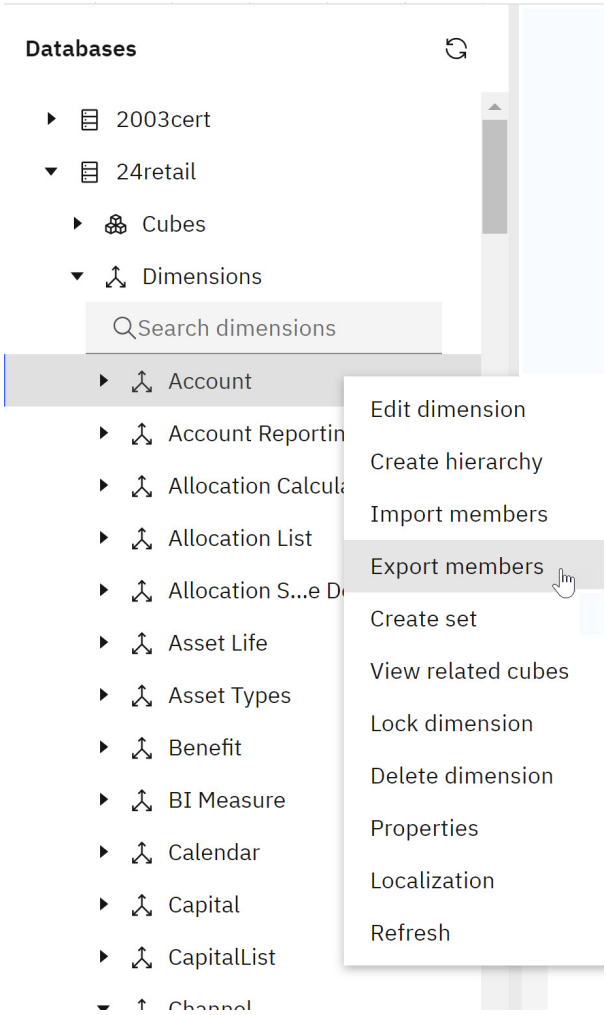

When you export data, you can set options that determine:

- which set and/or members to include in the export
- a range of numeric or string values to include in the export
- whether consolidated, rule-derived, or zero value are included in the export

## Export a dimension or hierarchy from a modeling workbench

You can now export a dimension or a dimension hierarchy to a `.csv` file. The exported file is saved to your local `Downloads` directory.

To export, right-click a dimension or hierarchy on the modeling workbench **Databases** tree, then click **Export members**.




You can also click the **Export members** icon on the dimension editor to export a dimension.

When you export a dimension or hierarchy, the resulting `.csv` file includes the following information about each member:

- name
- parent/child relationship
- type
- weight
- any defined attribute names and values

  Attribute names are prefixed with either ATTR_N or ATTR_S when they are exported, depending on the attribute type (numeric or string, respectively).

## Manage users and groups on a local Planning Analytics database

You can now manage Planning Analytics local database users and groups from the modeling workbench.

This capability requires that your Planning Analytics local environment uses TM1 authentication (as opposed to CAM authentication). Your authentication mode is determined by the **PAAuthMode** setting in the `paw.ps1` (Windows) or `paw.env` (Linux®) configuration file for your local installation.

To manage users and groups on a Planning Analytics database, right-click the database on the modeling workbench **Databases** tree, then click **Manage users and groups**. The procedures for managing users and groups on a local environment are the same as for Planning Analytics on Cloud. For more information, see Manage users and groups on a Planning Analytics database.

# What's new in administration - 2.0.99

The following new and changed administration features are available in Planning Analytics Workspace 2.0.99.

## IBM watsonx.ai integration and multiple deployment capabilities (Local only)

The IBM Cloud Pak® for Data integration, which supports Decision Optimization, was renamed to IBM watsonx.ai™ in Planning Analytics Workspace Local 2.0.99. You can now also configure multiple deployments of the IBM watsonx.ai integration. Both these changes are also available in Planning Analytics Workspace on Cloud 2.0.98 and later versions.

The **Integrations** tile now looks slightly different to support the ability to configure multiple deployments of the watsonx.ai integration. To add a new deployment, click **Add** or use the context menu of an existing deployment to duplicate it and edit the relevant fields.

You can modify or delete an existing deployment and use the **Status** toggle to enable or disable a deployment.

For more information, see Configure integration with IBM watsonx.ai to support Decision Optimization.

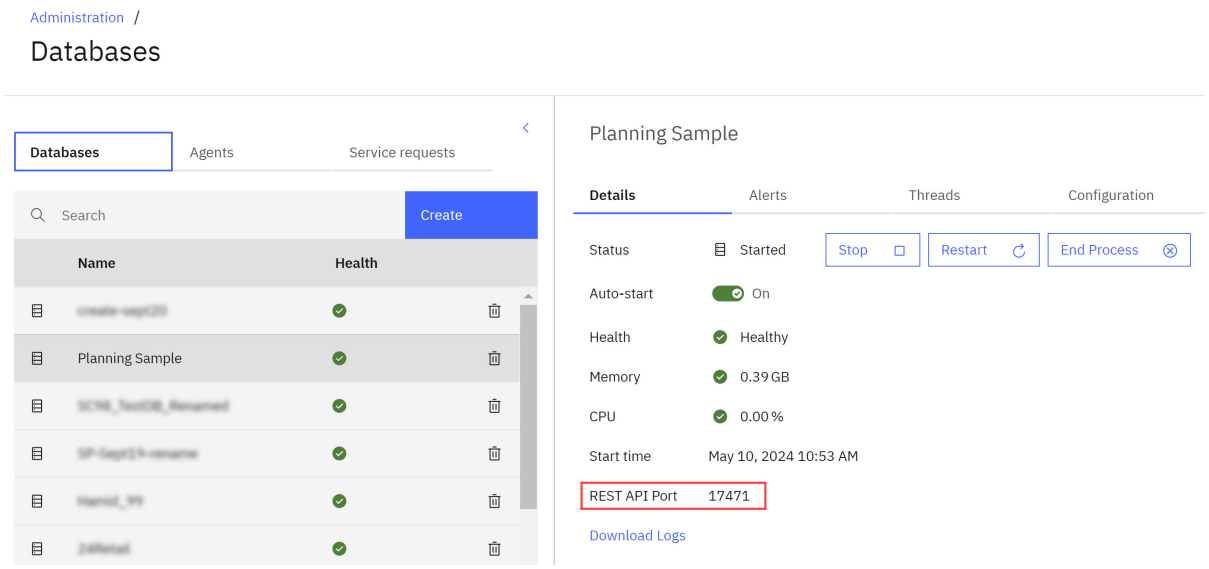
## Identify the HTTP port number for a Planning Analytics database (Cloud only)

You can now easily identify the HTTP port number for a database in Planning Analytics Administration.

To view the port number for a database:

1. Open Planning Analytics Administration.
2. Click the **Databases** tile, then click a database name.

   The port number is shown as the **REST API Port** on the **Details** tab.




   The port number is read-only; you cannot set the port number in Planning Analytics Administration.

# Known issues - 2.0.99

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

### Chart insights don't generate for views with summary calculations

An error displays when you try to generate insights for charts when the source view contains summary calculations on the columns.

1. Right click any column member in a view and select **Summarize all** to create a summary calculation.
2. Convert the widget to a Column chart.
3. Click on **Chart insights** in the toolbar.

Instead of generating insights, an error displays.

## 2.0.98 - What's new, August 21, 2024

Learn about new features and known issues in version 2.0.98 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.99

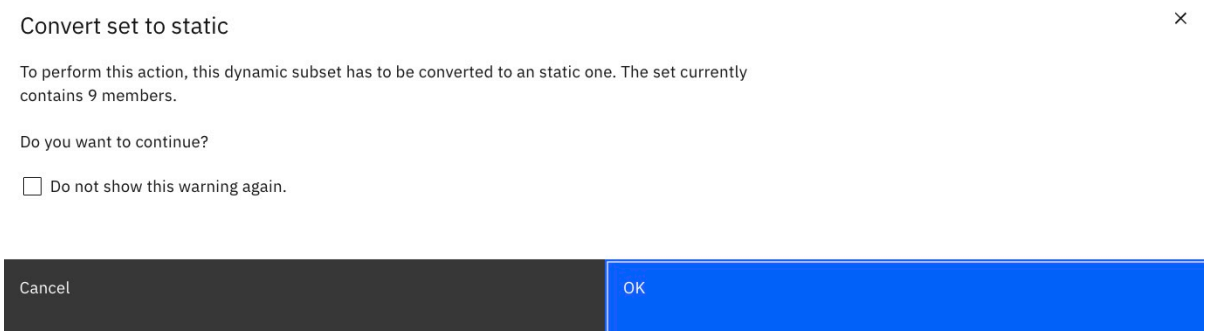
The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.99.

### Dynamic to static set conversion when re-ordering members

When you move members in the Planning Analytics Workspace set editor, the set will be converted from a dynamic set to a static set.

When you convert to a static set, the MDX expression that generated the dynamic set is deleted and the set contains only the members that are present at the time of conversion.

When you re-order members in the **Current set**, or move a member from **Available members** into a dynamic set in the **Current set**, the following dialog box appears:




If you click **OK**, the members are converted to a static set and moved to the new location.

If you click the check box beside "Do not show this warning again", the dialog box won't appear when you move members. This input is only valid for the current user session.

To disable and enable this dialog box across all user sessions, navigate to **Settings** > **General** > **Show dialog notification**, and toggle the **Show dialog notification** option.

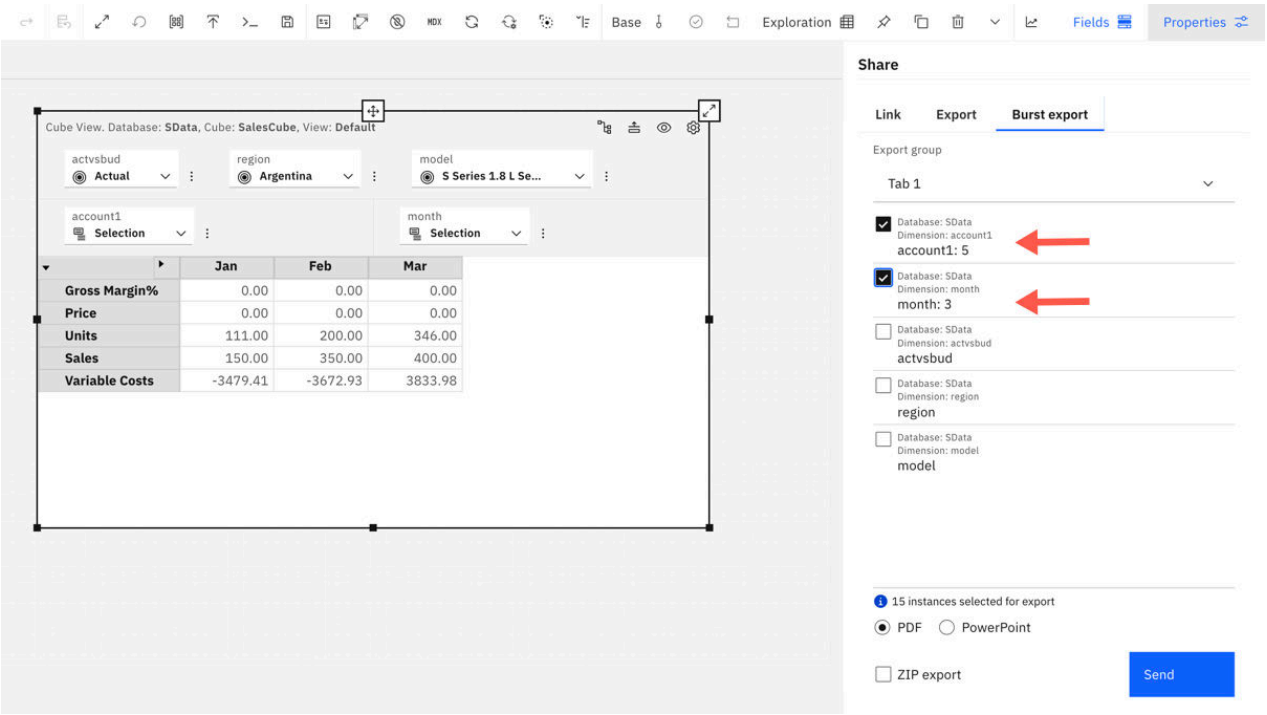
## New burst export feature

You can now slice a book in Planning Analytics Workspace and burst export it to PDF or PowerPoint by using single synchronization group. With this **Burst export** feature, you can automatically print each member of the set for a selected hierarchy to a PDF or PowerPoint file.

To use the **Burst export** feature, follow these steps:

1. In **Edit** mode, select the view and enable hierarchy synchronization (**Properties > Synchronization > Synchronize hierarchies**).
2. Click **Share** in the toolbar and go to the **Burst export** tab.
3. Select the group that you want to export under **Export group**.
4. Select the hierarchies that you want to export.

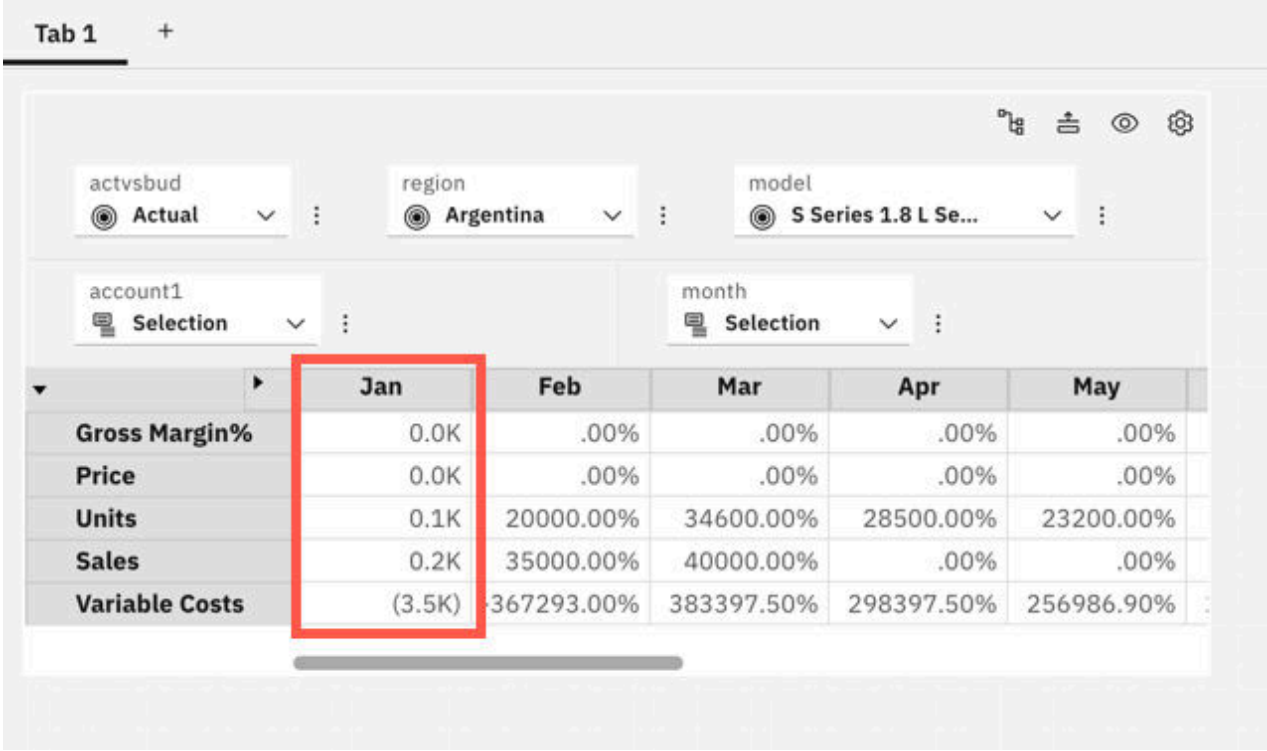
   **Note:** When you select a hierarchy, the number of members in that hierarchy displays under the hierarchy name. If you select more than one hierarchy, the total number of instances that export are calculated based on all the data combinations for the selected hierarchies. You cannot export more than 100 instances.
5. Select the file type that you want to export to. If you want to export individual files for each instance, select **ZIP export**.




## Improved application of member numeric formats

Member numeric formats are now retained when a numeric format is applied to the **Data area** in a view using the **Format manager**. Previously, applying numeric formats to the **Data area** of a view overrode the numeric formats of members.

In this view, notice *Jan* has a different numeric format from the other members in the view.




To change the numeric format for a member, right-click the member in a view and select **Member format**. Under **Format data cells**, select a numeric format.

To apply a global numeric format to the data cells in a view, select the view and click **Format manager**. Click **Data area** and select a numeric format.

## More actions are now available in the Planning Analytics AI assistant

The Planning Analytics AI assistant in Planning Analytics workspace now supports three new actions and can perform several additional member calculations.

The three new actions are List, Undo, and Redo. The additional member calculations are indicated in bold in the following table.

| Action | Definition of action | Command examples | Needs object/ widget selected? |
|---|---|---|---|
| List | Shows all the specified objects on a database. You can list cubes, dimensions, sets, and views. | `List cubes`<br>`List dimensions` | No |
| Undo | Undo a book action. | `Undo` | No |
| Redo | Redo a book action. | `Redo` | No |
| Perform member calculation | Perform calculations on specified members.<br>The supported calculations are:<br>• Absolute value<br>• Aggregate<br>• Average | `Calculate average for members "Jan", "Feb"`<br>`Create maximum for members "Q1", "Q2"`<br>`Calculate minimum for members "Q1", "Q2"` | Yes |

| Action | Definition of action | Command examples | Needs object/ widget selected? |
|---|---|---|---|
| | • **Division**<br>• Maximum<br>• Median<br>• Minimum<br>• **Multiplication**<br>• **Percent change**<br>• **Percent of**<br>• **Percent parent**<br>• **Percent total**<br>• Rank<br>• **Subtraction**<br>• **Summation** | | |

For more information, see Use the Planning Analytics AI assistant to interact with books using natural language.

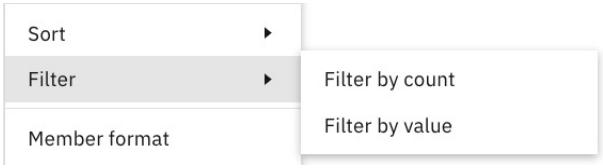
## Use numeric filters in the cube viewer

You can now filter rows and columns based on cell values in a cube in Planning Analytics Workspace.

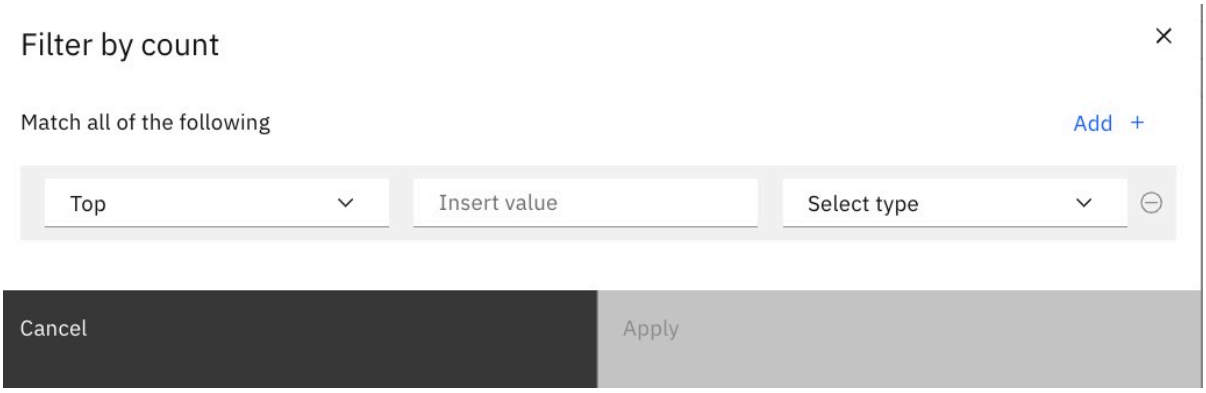
Two new filters have been added to Planning Analytics Workspace: **Filter by count** and **Filter by value**.

To use these new filters, right-click on a row or column header, and select **Filter**. Then select **Filter by count** or **Filter by value**.

**Note:** You can only have one **Filter by value** or **Filter by count** filter active at the same time. If you apply another filter of the same type to the same view, the first filter will be removed.



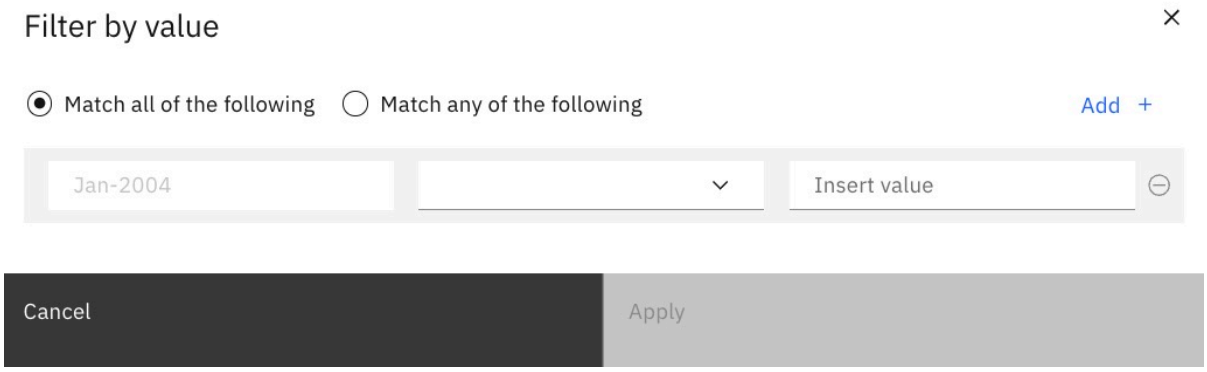

Filter by count to filter members in a row or column to show the top or bottom members of a specified count type.




You can choose the following criteria when creating the filter:

- You can select if you want the filter to show the top or bottom members of your specified count type.
- You can choose the number of members you want to see included in the filter results. This value must be numeric.
- You can select the type of count you want to filter for. You can choose **Items**, **Percent**, or **Sum**.
- You can add an additional set of criteria for the filter. You can add a total of two sets of criteria in each filter.

Filter by value to filter the members in a row or column based on a selected value.




You can choose the following criteria when creating the filter:

- You can select a comparison operator for the filter. You can choose **=** (equal to), **>** (greater than), **>=** (greater than or equal to), **<** (less than), or **<=** (less than or equal to).
- You can choose the number you want the comparison operator to filter by. This value must be numeric.
- Select how you want the filter to match your criteria. Select **Match all of the following** if you want the filter to only include members that match all of the criteria you set. Select **Match any of the following** if you want the filter to include members that match any of the criteria you set.
- You can add an additional set of criteria for the filter. You can add a total of three sets of criteria in each filter.

A row or column with a filter applied has the filter icon .

You can use filters and sorting capabilities simultaneously. To learn more about sorting, see Sort rows and columns.

For more information on filtering rows and columns, see Filter rows and columns based on cell value.

# What's new in modeling - 2.0.98

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.98.

## Lock and unlock an element

You can now lock and unlock an element in IBM Planning Analytics Workspace.

When a user locks an element, only those users who have Admin rights for that element can update the data that it identifies. Even the user who locks the element cannot update its data, unless they have Admin rights for that element.

Locking an element is a way to permanently archive the data that it identifies. Locks stay in place after the remote server shuts down.

**Note:** To use this feature, IBM Planning Analytics 2.0.9.19/TM1 11.8.023 is required.

For more information on how to lock and unlock an element, see Lock and unlock an element in IBM Planning Analytics Workspace

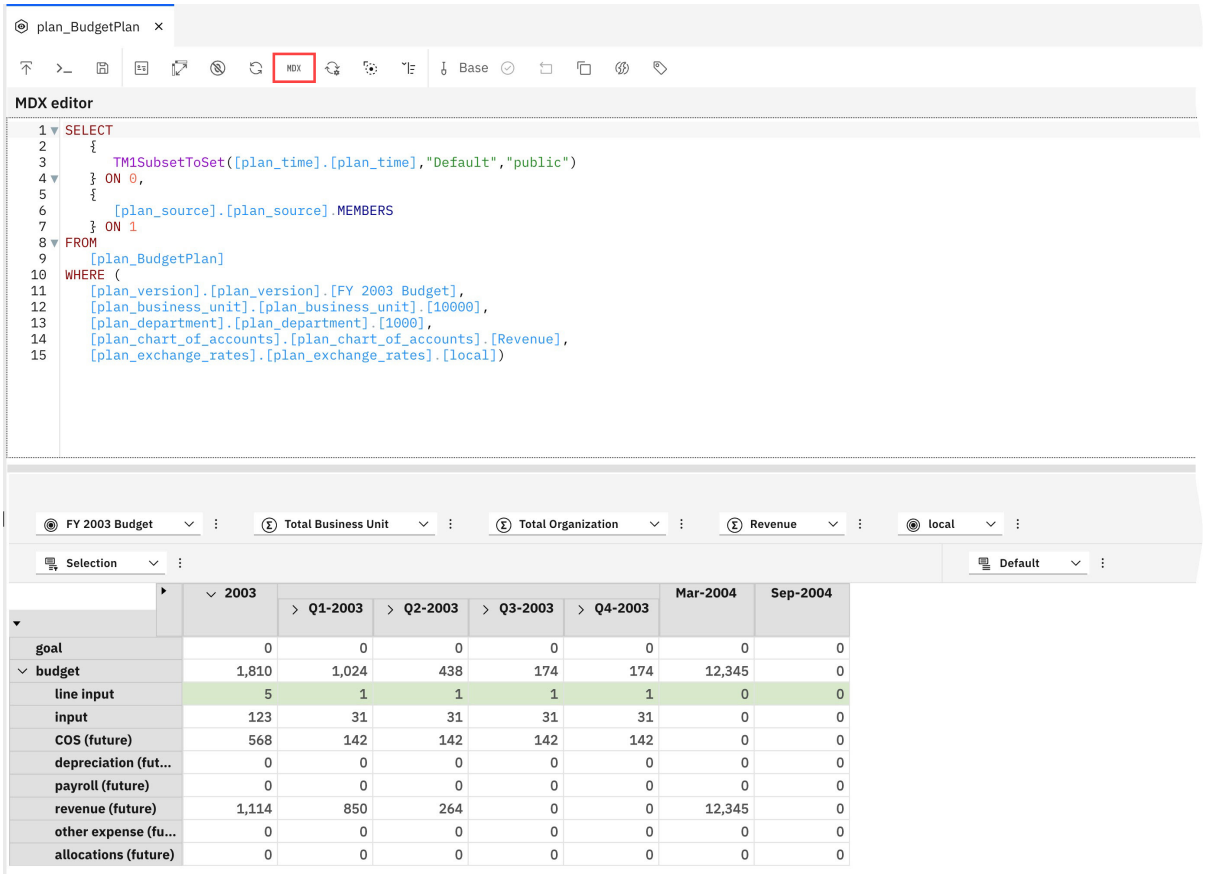
## MDX editor updates

A new MDX editor is now available in the IBM Planning Analytics Workspace books and modelling workbench.

**Note:** The new MDX editor is not currently available in IBM Planning Analytics for Microsoft Excel.

The MDX editor allows you to make changes to the MDX expressions in a view.

To access the MDX editor, click the **Edit MDX** icon MDX .




| | 2003 | Q1-2003 | Q2-2003 | Q3-2003 | Q4-2003 | Mar-2004 | Sep-2004 |
|---|---|---|---|---|---|---|---|
| goal | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| budget | 1,810 | 1,024 | 438 | 174 | 174 | 12,345 | 0 |
| line input | 5 | 1 | 1 | 1 | 1 | 0 | 0 |
| input | 123 | 31 | 31 | 31 | 31 | 0 | 0 |
| COS (future) | 568 | 142 | 142 | 142 | 142 | 0 | 0 |
| depreciation (fut... | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| payroll (future) | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| revenue (future) | 1,114 | 850 | 264 | 0 | 0 | 12,345 | 0 |
| other expense (fu... | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| allocations (future) | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

Several updates that have been made to the MDX editor:

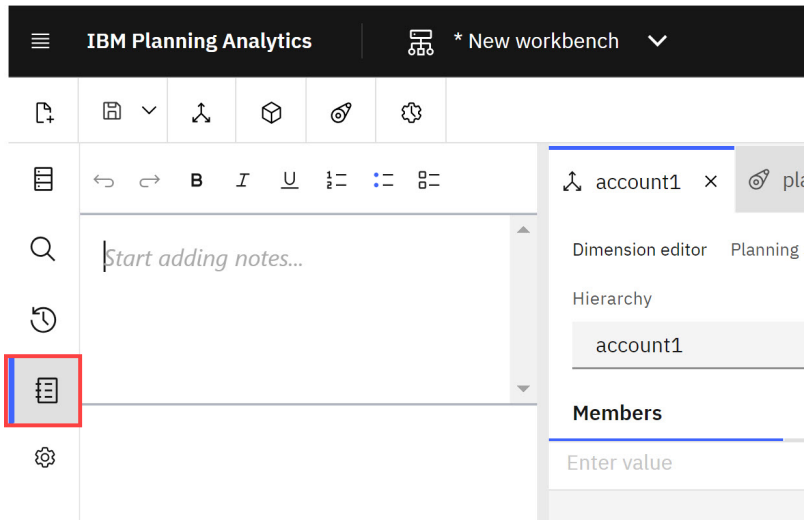
- Autocomplete: provides suggestions on dimension names, functions, and other TM1 objects while you are typing.
- Syntax coloring: shows different portions of the MDX statement in different colors for increased readability. For example, keywords are written in red, and tuples are written in blue.
- Undo: allows you to undo the last change.
- Redo: allows you to redo the last change.
- Find and replace: clicking CTRL+f open a dialogs that allows you to search for specific text in the MDX, and replace it.
- Indenting: the MDX editor now automatically formats the MDX script, indenting each line. When you make changes in the view, the MDX editor updates with the correct formatting. You can manually indent the lines in the script by clicking **Indent script** in the MDX editor toolbar.
- **Apply** applies your MDX edits to the view. When you click **Apply**, a text box appears telling you if the changes were made successfully.
- **Synchronize** replaces the MDX in the editor with the one represented in the view. After you click **Synchronize**, click**OK** to synchronize.

**Note:** When you make edits in a view, the MDX editor automatically synchronizes to match these changes. However, when you make edits in the MDX editor, the view does not automatically synchronize to match these changes. You have to manually select where you want your edits to apply using the **Apply** and **Synchronize** buttons in the MDX editor toolbar.

## Add notes to a modeling workbench

You can now add notes to a modeling workbench to keep track of your work, document the purpose of an object or workbench, or instruct team members on the use of a workbench.

To create or edit a note, click the **Note** icon on the side toolbar of the workbench.




Enter your note text and use the formatting toolbar to apply text formats or create lists. If you enter a valid URL in a note, it displays as a clickable link. Click **Save** to save the note to the workbench.

## Use the new Drive explorer to access the S: drive (Cloud only)

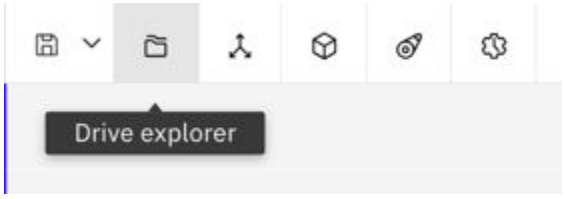
You can now access the S: / drive through the Planning Analytics Workspace modeling workbench with the new **Drive explorer**.

The Drive explorer is not available in IBM Planning Analytics as a Service.

The **Drive explorer** option is visible to users with Subscription administrator access or those for whom the **Drive explorer** capability is enabled.

Users with access to **Drive explorer** have full access to the file system in the S: / drive and the remote server. There is no access control beyond enabling the **Drive explorer** capability.

Click **Drive explorer** to open the root of the S: / drive.




Like in the **File manager**, you can upload files, create folders, and select files in the **Drive explorer**. Selected files can be renamed, moved to a new location, copied, deleted, and compressed.

The **Drive explorer** has an additional capability that lets you connect to the remote server associated with your Cloud. Once connected, you can move files to a new location.

1. Click **Open remote server** from the **Drive explorer** toolbar.




2. Log in to your remote server with your Fileshare credentials and click **Refresh folder items** to test the connection. The credentials to connect to the remote server are provided in the welcome kit for that specific Planning Analytics system.

You can now see the files on your remote server.

You can create folders here and organize files and select files to rename, delete, and move them to a new location. Keep in mind, when you copy a file from the remote server, you can paste the file to the local `S:/` drive only. You can also copy files from the `S:/` drive and paste them to the remote server.

### Known issue in the initial release of Drive explorer

You should be aware of this issue that you might encounter while using the Drive explorer in Planning Analytics Workspace 2.0.98.

**Error when uploading and expanding a .zip archive when the file extension is all upper case**

If an archive file is saved with the `.ZIP` extension (using all upper case characters, as opposed to `.zip`), you receive an error when attempting to extract the archive after copying and pasting the archive from the Drive explorer to your local `S:/` drive.

To resolve this issue, rename the archive on the remote location using a lower case file extension before copying and pasting to the local `S:/` drive.

## What's new in administration - 2.0.98

The following new and changed administration features are available in Planning Analytics Workspace 2.0.98.

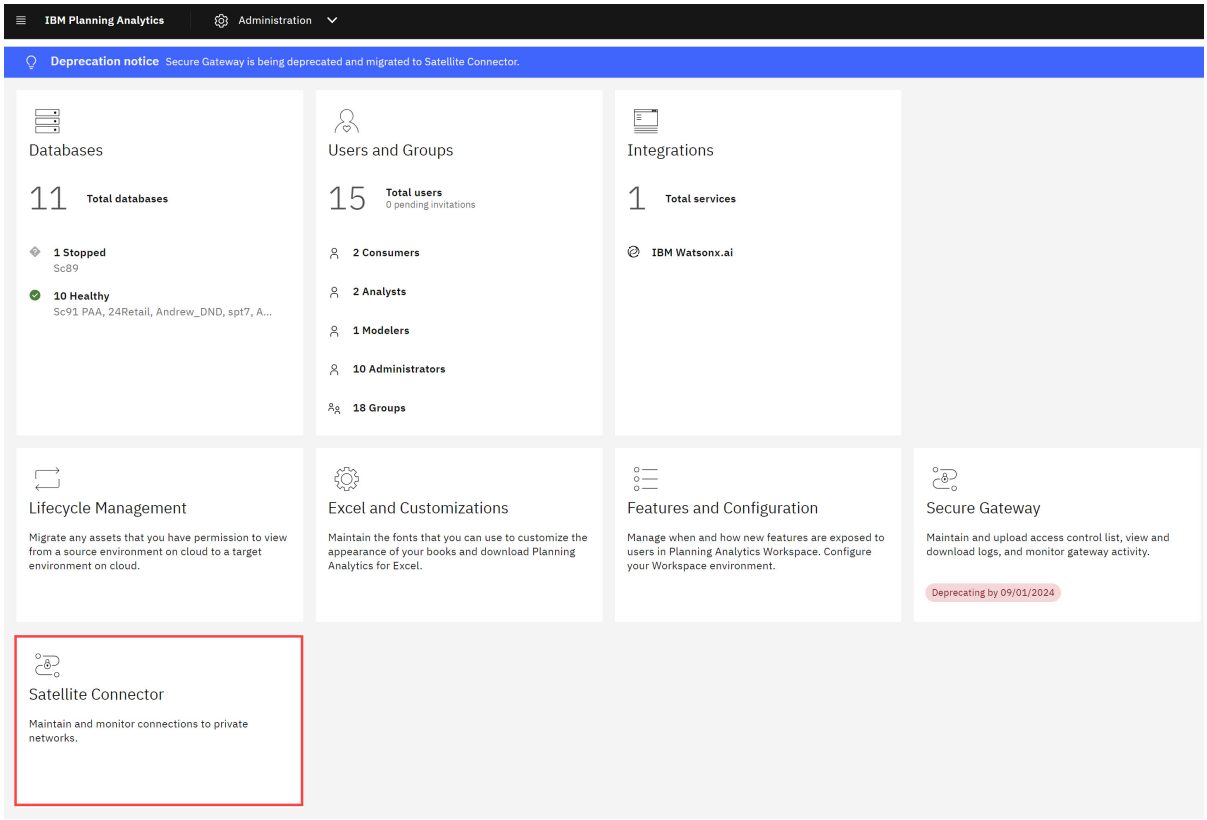
### Use the new Satellite Connector to connect to a private network (Cloud only)

IBM Satellite Connector is IBM's new and improved VPN tunneling technology, which provides a secure connection between IBM Planning Analytics on Cloud and a private network where your on-premise data resides.

Satellite™ Connector provides functionality similar to Secure Gateway, but Satellite Connector has no bandwidth egress limits, supports more concurrent incoming connections, includes support for the latest generation of VPC networking, and supports a greater number of client connections for client-side high availability.

Deprecation of Secure Gateway was previously announced online and in the Secure Gateway interface on Planning Analytics Workspace. Both Satellite Connector and Secure Gateway are available in Planning Analytics Workspace 2.0.98, but Secure Gateway will be removed from Planning Analytics Workspace in early October 2024. You can review this blog post to prepare for migration to Satellite Connector.

You can create and manage Satellite Connector connections from the Administration page in Planning Analytics Workspace.




Full documentation on using Satellite Connector and migrating from Secure Gateway to Satellite Connector will be available in Planning Analytics Workspace 2.0.98.

## Manage user capabilities in Planning Analytics Administration (Cloud only)

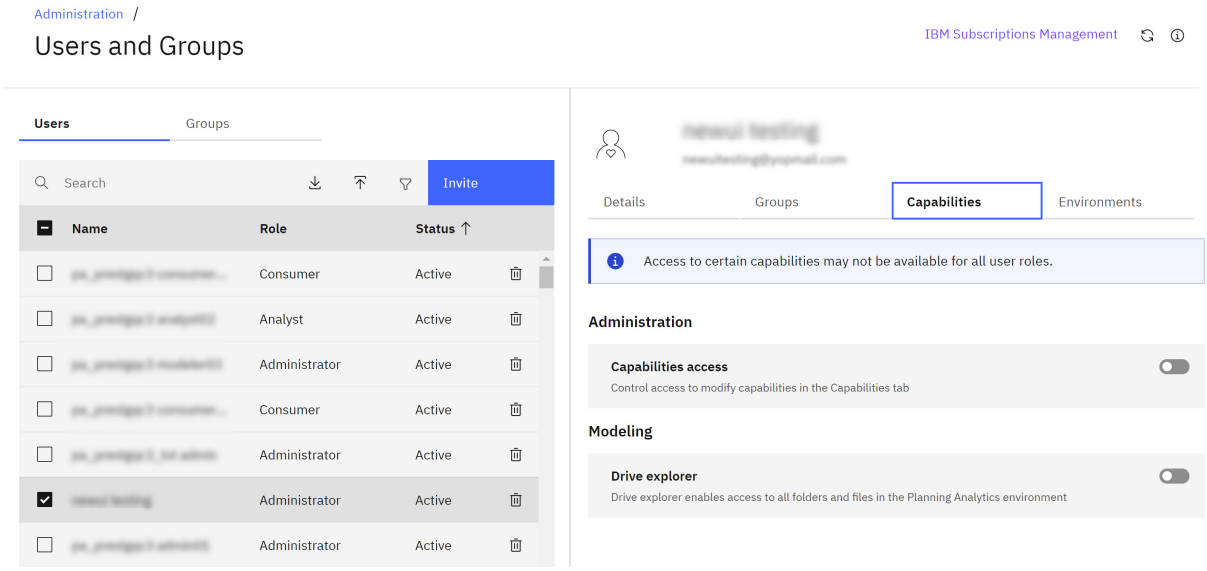
Administrators can now manage capabilities for individual users in Planning Analytics Administration.

For Planning Analytics Workspace 2.0.98, the most significant capability that can be managed is **Drive explorer**. This capability must be enabled for a user to access the S: / drive through the Planning Analytics Workspace modeling workbench.

Initially, only a subscription administrators can manage capabilities and only in a primary environment. However, a subscription administrator can assign the ability to manage capabilities to other administrators in either a primary environment or in a non-primary environment. See the Planning Analytics Workspace 2.0.98 documentation for full details on managing capabilities.

To manage capabilities for a user:

1. Open Planning Analytics Administration as a subscription administrator.
2. Click the **Users and groups** tile.
3. Click the **Users** tab.
4. Select the user for whom you want to manage capabilities.
5. Click the **Capabilities** tab.




The tab shows the capabilities that can be managed. In Planning Analytics Workspace 2.0.98, there are two capabilities that an administrator can manage.

**Capabilities access**
: Enable to grant an administrator the ability to manage capabilities for users. If this capability is not enabled for an administrator user, the administrator cannot see the **Capabilities** tab and cannot manage capabilities.

**Drive explorer**
: Enable to give the user access to the **Drive explorer** on the modeling workbench. The **Drive explorer** provides access to all folders and files in the Planning Analytics environment.

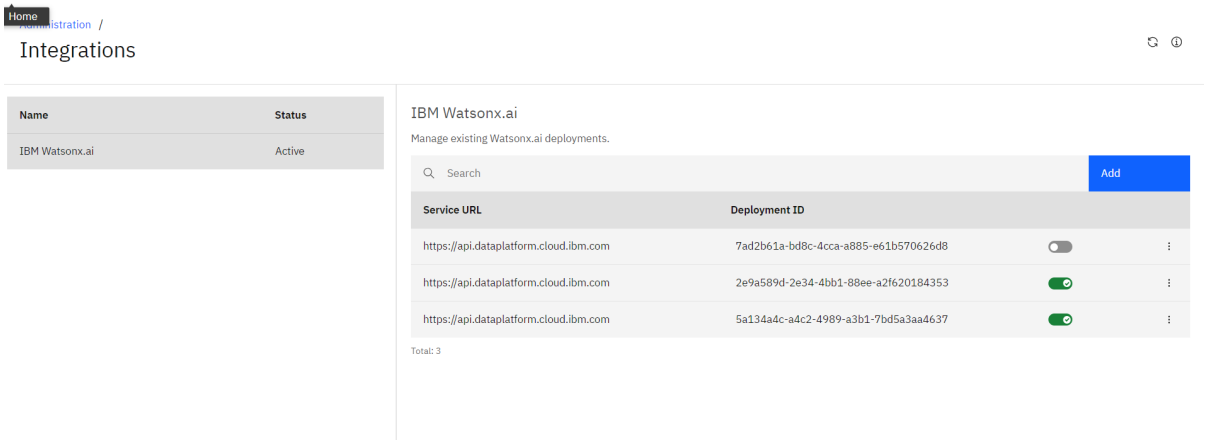
6. Click the toggle to enable or disable a capability for the user.

## IBM Cloud Pak for Data integration renamed and multiple deployment capabilities added

The IBM Cloud Pak for Data integration, which supports Decision Optimization in Planning Analytics Workspace on Cloud, was renamed to IBM watsonx.ai in this release. Along with this change, you'll notice that you can now configure multiple deployments of the IBM watsonx.ai integration.

The **Integrations** tile now looks slightly different to support the ability to configure multiple deployments of the watsonx.ai integration. To add a new deployment, click **Add** or use the context menu of an existing deployment to duplicate it and edit the relevant fields.

Use the **Status** toggle to enable or disable a deployment.




You can modify or delete an existing deployment through its context menu.

# 2.0.97 - What's new, July 16, 2024

Learn about new features and known issues in version 2.0.97 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## General updates - 2.0.97

The following new and changed features are available across Planning Analytics Workspace 2.0.97 and do not apply to any single functional area of the release.

### Define workspace settings from a central Settings editor

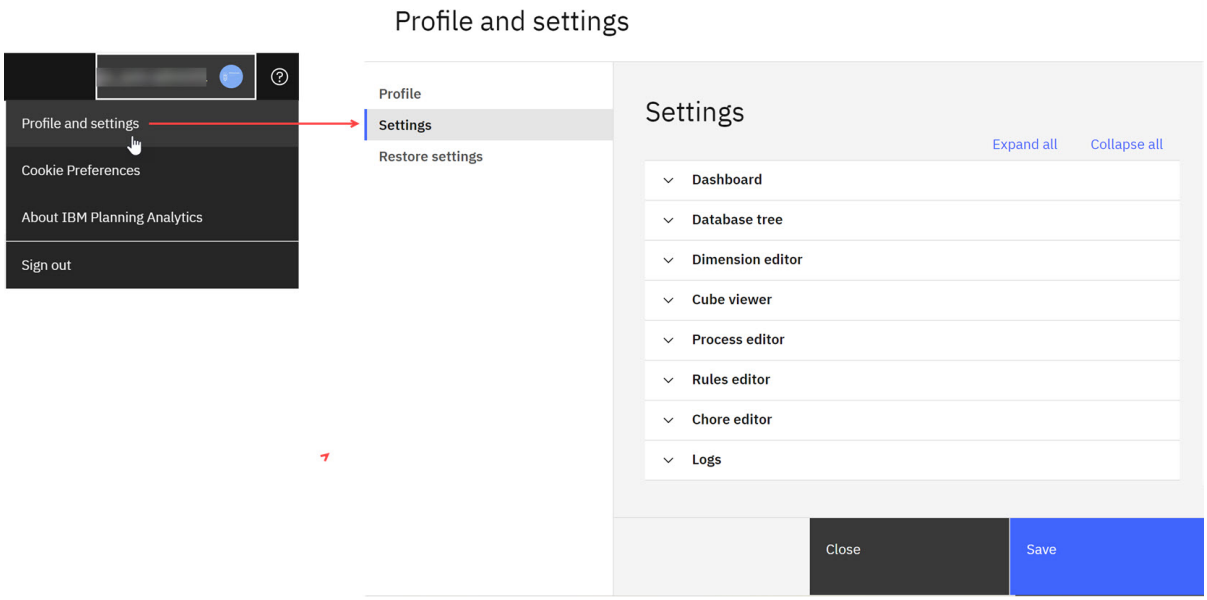
Administrators, Modelers, and Analysts can now use a single Settings editor to define settings and preferences for multiple Planning Analytics Workspace components and features.

The Settings editor is not available for Consumers.

While the Settings editor defines global settings for components and features, you can still modify settings directly in an instance of a component. For example, when using the dimension editor, you can click the **Settings** icon and modify the settings for that instance of the editor.

To use the Settings editor:

1. Click your user name on the Planning Analytics Workspace title bar.
2. Click **Profile and settings**.
3. Click **Settings**.
4. Click a settings category, then define your settings.



The components and features available in the Settings editor vary according to your user role in Planning Analytics Workspace. You'll see only the components and features for which you can set preferences. For example, an Administrator sees all components and features, but an Analyst sees only **Dashboard**, **Database tree**, **Dimension editor**, and **Cube viewer**.

You can also reset settings to default values from the same location:

1. Click your user name on the Planning Analytics Workspace title bar.
2. Click **Profile and settings**.
3. Click **Restore settings**.
4. Click the category for which you want to restore settings to default.

# What's new in books and reporting - 2.0.97

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.97.

## Action button references now updated when you copy and paste books to a new folder

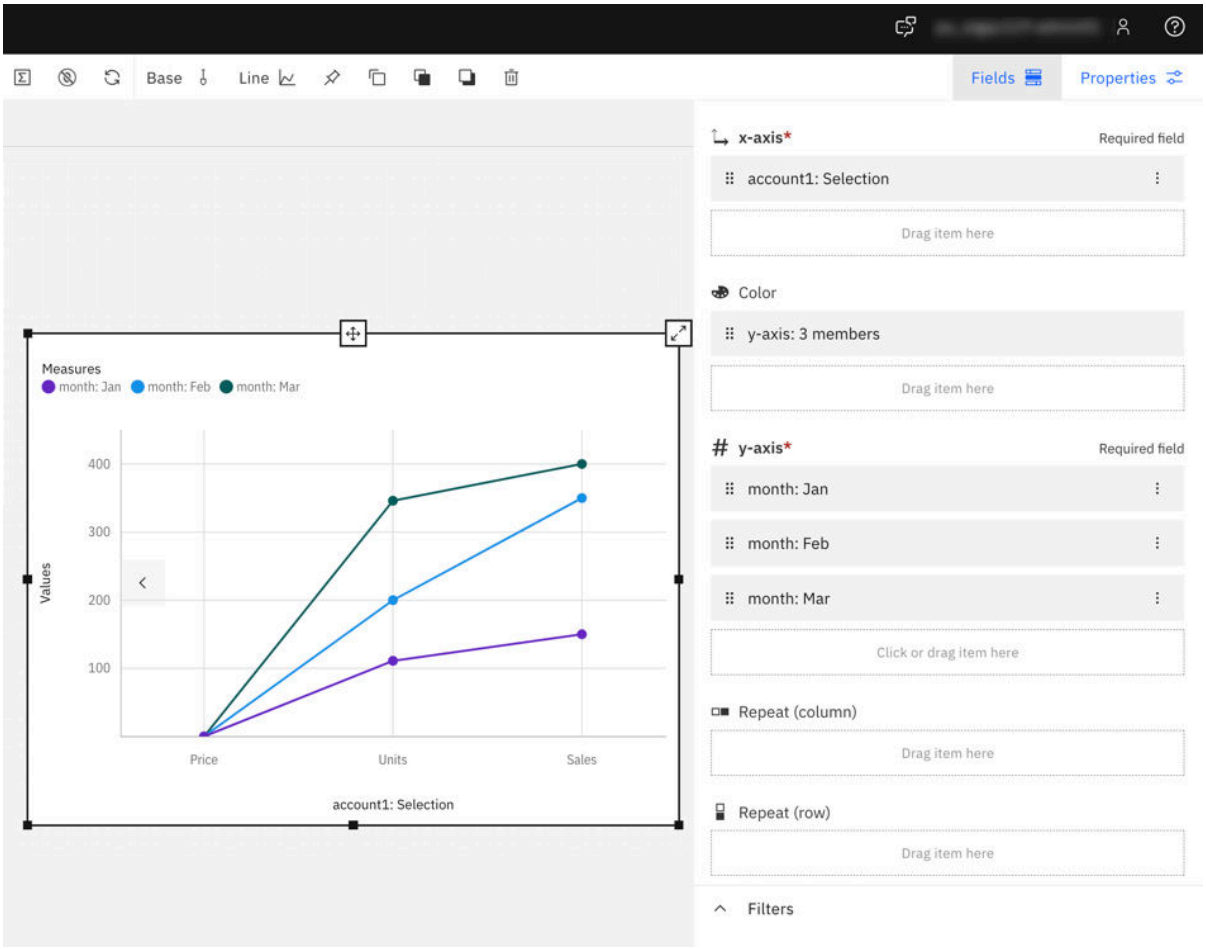
When you copy and paste multiple books from one folder to a different folder, action buttons in the books that reference any other books *which are being copied at the same time* are automatically updated. The action buttons are adjusted to point to the new location of a copied/pasted book, not to the original location.

If you copy and paste a book to a new folder, but the book contains an action button that references another book in the original folder, the action button continues to reference the book in the original folder.

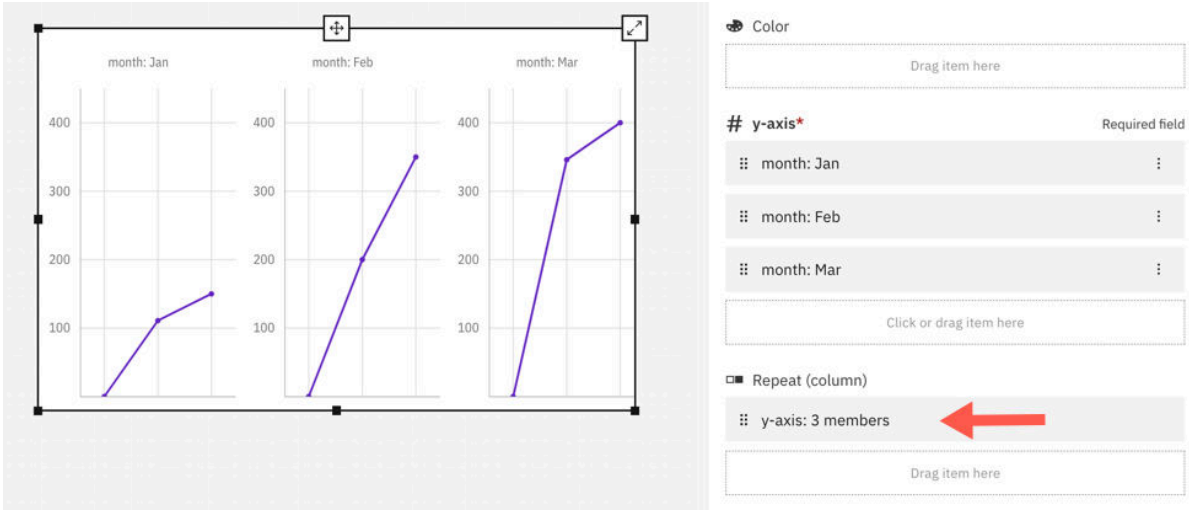
## Plot members from the same dimension in certain charts

Certain visualizations can now plot members from the same dimension, which makes it easy to view and compare members quickly.

In the following example, all members of the *month* dimension display in the **Line** chart as measures on the y-axis.



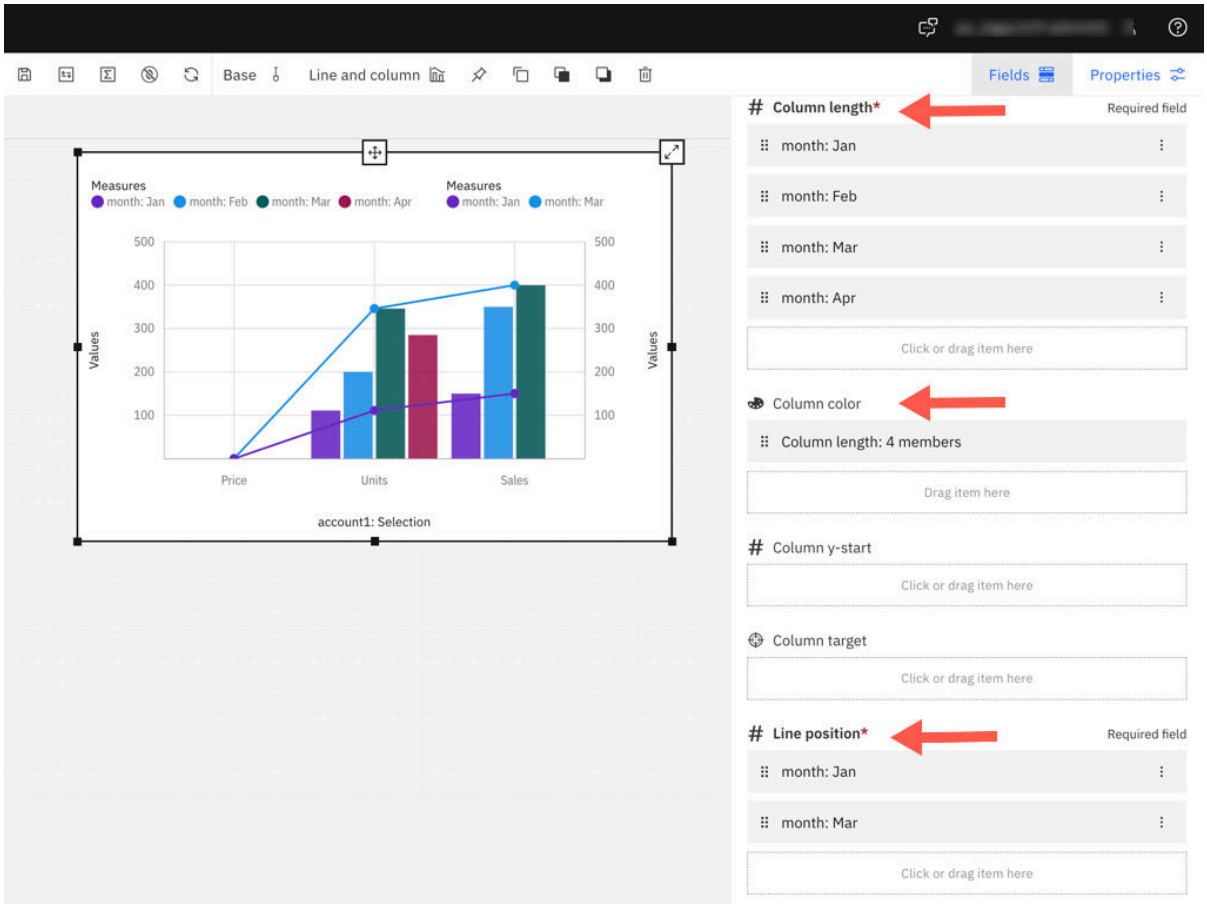

Notice in the **Fields** panel, the **Color** field shows an item that is called **y-axis: 3 members**, which shows that there are three members that display in different colors on the y-axis. This item in the **Color** field acts like any other hierarchy. You can drag it from the **Color** field to other fields like the **Repeat (column)** to see individual charts for each member.




You can click into an empty y-axis field to plot another member. To remove an existing member, click the menu icon for the member you want to remove and select **Remove**. When the set changes, the y-axis field also updates to show the members in the new set.

Similarly, when you create a combination visualization, members of the column dimension are plotted on the **Column length** or **Length** fields and the **Color** field displays the set of members.

In the **Line and column** chart, members of the column dimension appear on both the **Column length** and **Line position** fields. You can then choose which members display as columns and which member should be a line.




The following visualizations can plot multiple members from the same dimension:

- Column
- Line
- Line and column
- Bar
- Stacked bar
- Stacked column
- Marimekko
- Point
- Area

## Refresh the current set and MDX query without making changes

You can now refresh the **Current set** and the MDX query without making any changes to the set or the MDX string.

The **Reset** button for the **Current set** in the set editor now also allows you to refresh the set even if there are no changes to the set. Similarly, you can use the **Commit** button in the MDX to refresh the query without changing the MDX string.

Previously, both the **Reset** and **Commit** buttons were disabled until changes were made.

### Format spacer cells in the Format manager

You can now apply formatting to the spacer cells in a view using the **Format manager**.

Open the **Format manager** from toolbar and under the **View** tab, click **Spacers** to set the background color or to show/hide the line dividers for spacer cells.

You can also modify existing spacer cells, by right-clicking the spacer cells in a view and selecting **Spacer format**.

**Note:** This release also includes feature enablement for the **Format manager**. For more information about using the **Format manager**, see Format cube views with the Format manager.

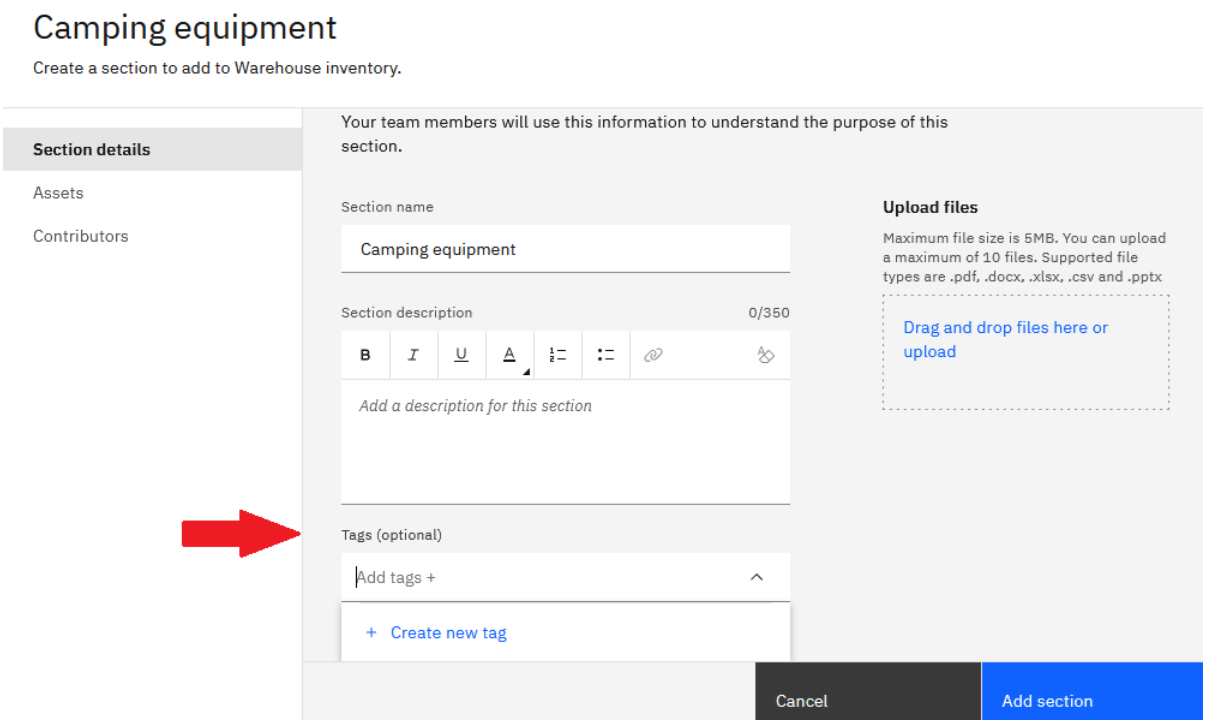
## What's new in plans and applications - 2.0.97

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.97.

### Create color-coded tags in plans and applications

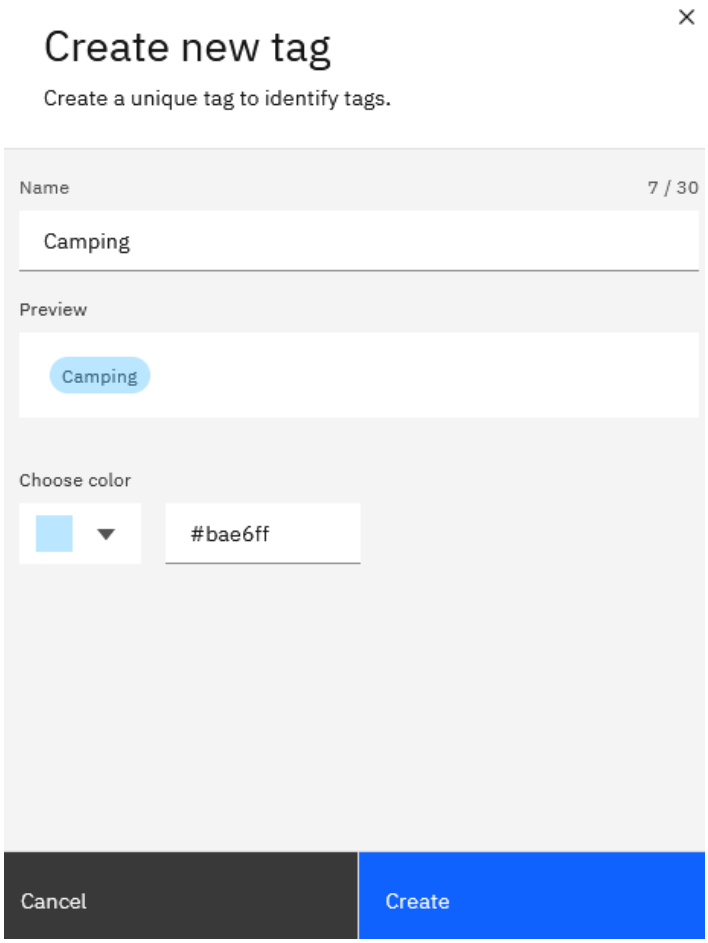
Color-coded, custom tags in plan tasks and application sections help you to quickly identify tasks or sections on the Overview page.

You can create tags on the **Details** tab when creating or editing a task or section.

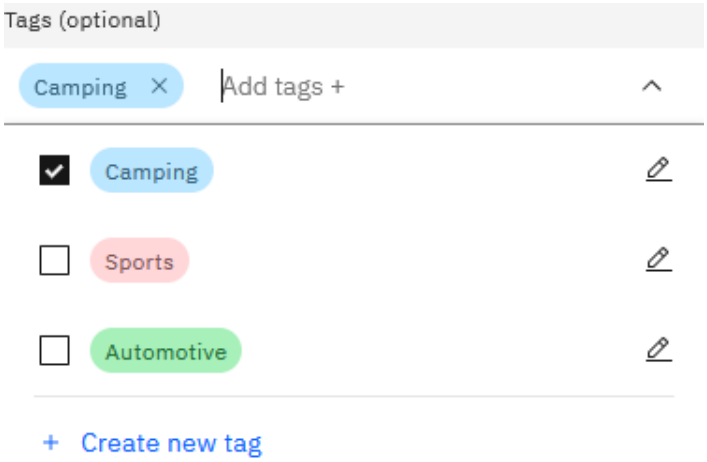



To create and add a new tag to your task or application, follow these steps:

1. On the **Section details** or **Task details** tab, click into the **Tags** field, then click **Create new tag**.
2. In the **Name** field, enter a label for your tag.
3. Choose a color for your tag. You can see what the tag will look like in the **Preview** field.




4. Click **Create**.
5. Expand the **Tags** field, select the tag that you just created, and click **Save** to add it to the task or section.




Any tags that you create are saved to the plan or application. This means that all tasks in a plan or all sections in an application can share the available tags in the same plan/application, but can't access tags from other plans/applications.

Click the **Edit a tag** icon next to a tag to edit or delete the tag.

**Important:** When you delete a tag, the tag is removed from all tasks or sections that use the tag.

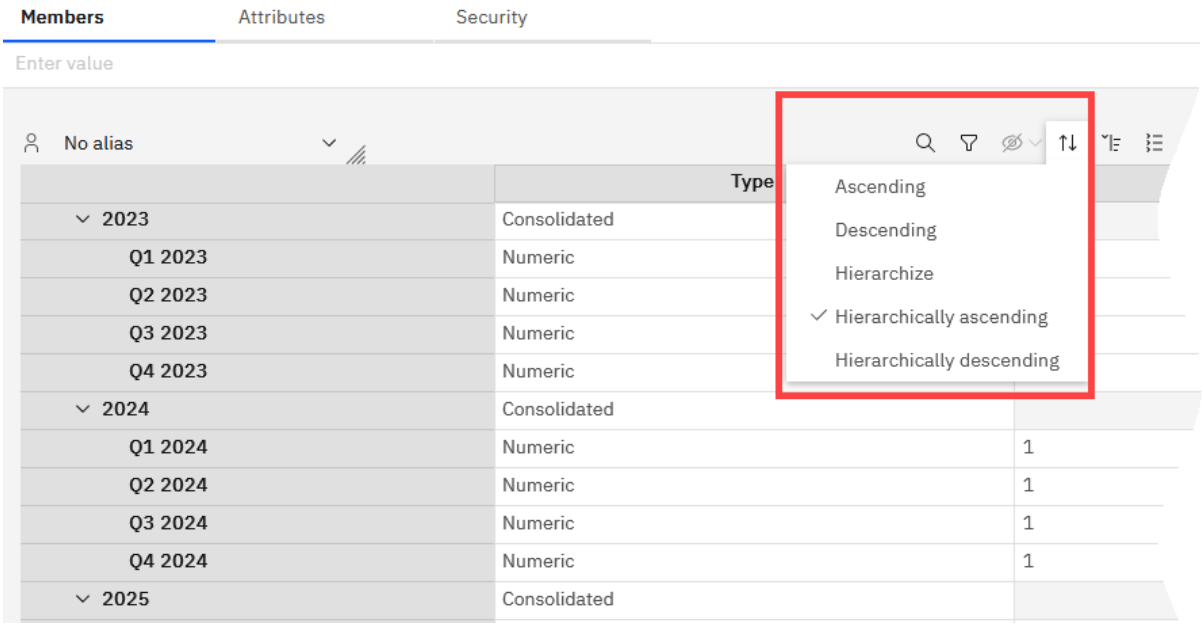
# What's new in modeling - 2.0.97

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.97.

## Sort hierarchy members in the dimension editor

You can choose from several sort orders to sort hierarchy members in the dimension editor.

Click the **Sort members** icon ↑↓ in the toolbar to access the sort options.




The following sort options are available:

**Ascending**
Sort members in ascending order.

**Descending**
Sort members in descending order.

**Hierarchize**
Organize the members of specified set into hierarchical order.

**Hierarchically ascending**
Sort members within a hierarchy in ascending order.

**Hierarchically descending**
Sort members within a hierarchy in descending order.

For more information, see Welcome to the new experience dimension editor.

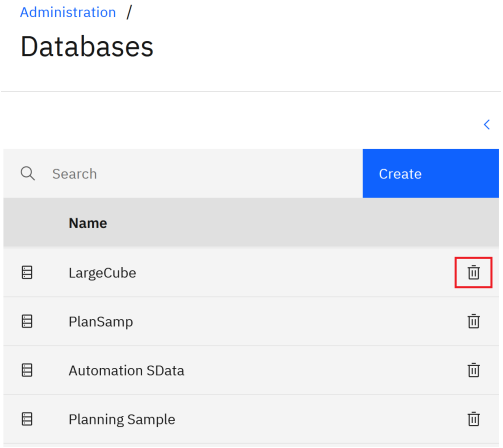
# What's new in administration - 2.0.97

The following new and changed administration features are available in Planning Analytics Workspace 2.0.97.

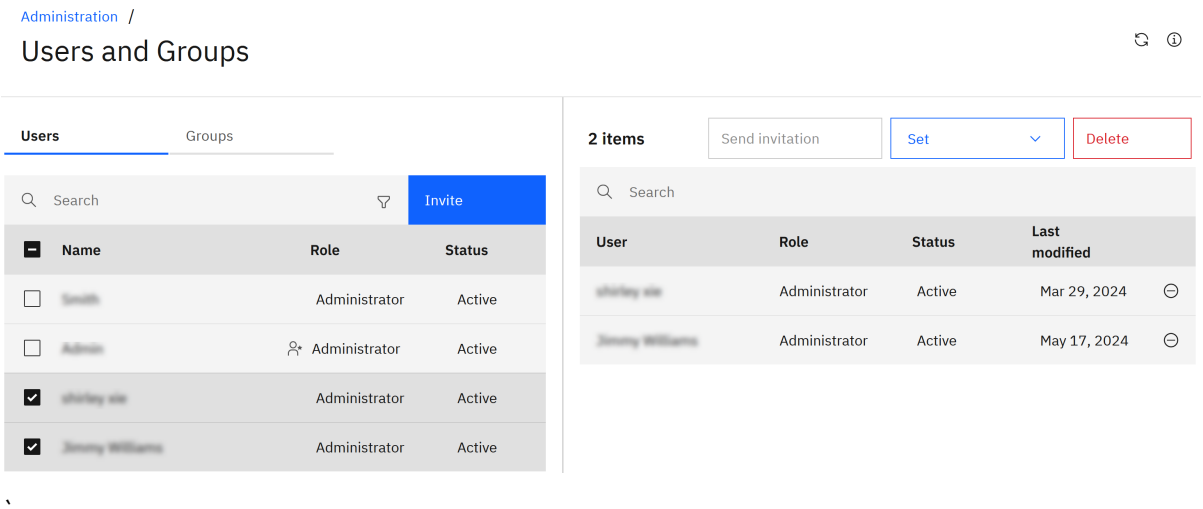
## Improved user interface for deleting objects

The delete icon in Planning Analytics Administration is now located next to each item name in all administration interfaces.

For example, the following image shows the delete icon next to each database name.




On the **Users and Groups** > **Users** tab, when you select multiple users, the **Delete** button appears in the right panel.




# Deprecation notices - 2.0.97

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.97.

## End of support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 (Local only)

Support for Planning Analytics Workspace Local on Red Hat® Enterprise Linux 7 will end in June 2024.

This coincides with Red Hat's announced end of maintenance support for Red Hat Enterprise Linux 7. For more information, see https://www.redhat.com/en/blog/end-maintenance-red-hat-enterprise-linux-7-almost-here.

Planning Analytics Workspace Local versions 2.0.70 and later are supported on Red Hat Enterprise Linux version 8.

Planning Analytics Workspace Local versions 2.0.93 and later are supported on Red Hat Enterprise Linux version 9.

# 2.0.96 - What's new, May 28, 2024

Learn about new features and known issues in version 2.0.96 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.96

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.96.

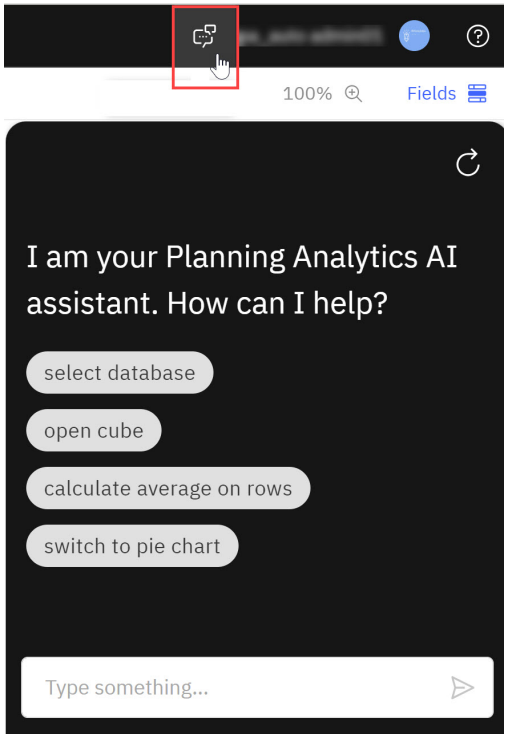
### Use AI to interact with books using natural language

The new Planning Analytics AI assistant lets you perform a wide variety of actions on books using simple natural language commands.

The initial release of Planning Analytics AI assistant supports English only. The Planning Analytics AI assistant is available only when working directly in a book. The assistant is not available when you open a book from a plan or application, nor is it available from a modeling workbench.

**Important:** Planning Analytics AI assistant requires a separate license purchase, it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

To use the assistant, open a book and click the **Planning Analytics AI assistant** icon on the toolbar, right next to your user name.



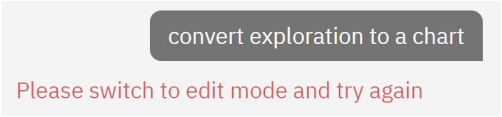
Type a natural language command or instruction in the Planning Analytics AI assistant. There are no hard and fast rules about how you issue a command, as long as the command includes a single action that you want to perform and (in most cases) the subject of the action. For example, if you want to swap the rows and columns in an exploration, all of these commands will work:

- `swap rows and columns`
- `switch rows`
- `exchange rows and columns`
- `swap` (notice there is no subject specified here)

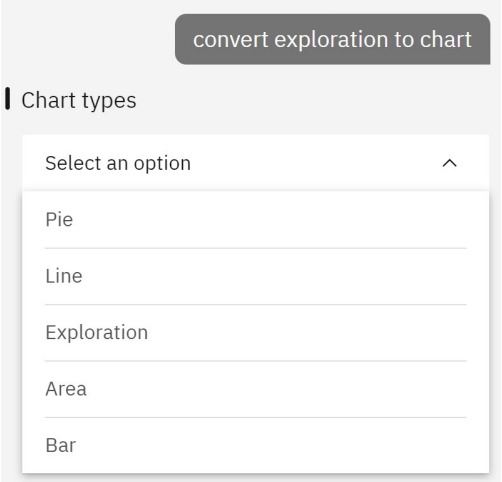
To reuse a previous command from the current session, place the cursor in the Planning Analytics AI assistant text entry box, then use the Up and Down keyboard keys to cycle through recent commands. When the desired command is in the text box, press Enter to use the command.

Some commands require that an object or widget on a book be selected before typing the command, others require that the book be in Edit mode before the command can be executed. When your command is ambiguous or cannot be completed in the current context, Planning Analytics AI assistant prompts you for clarification.

For example, if you try to convert an exploration to a chart, the assistant notifies you that you must be in Edit mode to perform this action.




Similarly, if you are in edit mode and command "`convert exploration to chart`" (a somewhat ambiguous request), the assistant prompts you to specify which type of chart you want and then completes your request.




**Tip:** If your command includes a reference to a specific object or artifact, it's helpful to enclose the item name in double quotes. This helps Planning Analytics AI assistant determine your intent and identify the item you want to work with.

Of course, you're not limited to creating charts. You can use Planning Analytics AI assistant to change the configuration of a view, open a view on a different database, open a dimension or set, perform member calculations, suppress zeroes, and more.

Remember, a command specifies a single action; you can't issue a compound command to perform multiple actions. For example, the command "`open the Revenue cube and suppress zeroes`" won't work.

Here's a "cheat sheet" to help you start using Planning Analytics AI assistant. It shows all the actions that you can perform with the assistant.

Notice that there are multiple ways to express most commands. The assistant recognizes synonyms for most actions, so you're not limited to one specific command construct. Experiment with commands, knowing that the assistant will prompt for clarification if a command is ambiguous.

| **Action** | **Definition of action** | **Command examples** | **Needs object/ widget selected?** |
|---|---|---|---|
| Select database | Set the search context for the Open actions in the book | `Select database 24Retail and Planning Sample`<br>`Work with "Planning Sample"`<br>`Set "24Retail" as database` | No |
| Show databases | Display the databases currently being searched | `What is my search context?`<br>`What is the current db?` | No |
| Open | Open one of these Planning Analytics objects:<br>• Cube (default view)<br>• Exploration (view)<br>• Dimension<br>• Hierarchy<br>• Set | `Open dim "Org"`<br>`Open "Asset Life" view`<br>`Show the "Marketing" set`<br>`Edit the "Employee" hierarchy` | No |
| Hide | Hide a member(s) from rows or columns | `Hide "Jan", "Feb"`<br>`Don't show the "Overhead" member` | Yes |
| Keep | Keep only the specified member(s) on the view | `Keep "Jan", "Feb"`<br>`Members to keep: "Washington", "California"` | Yes |
| Unhide all | Display all hidden members | `Unhide all`<br>`Show all rows`<br>`Uncover columns` | Yes |
| Switch visualization | Switch the visualization to a different type | `Switch to pie chart for "sales"`<br>`Convert to chart`<br>`Switch to exploration` | Yes |
| Summarize all | Create a calculation of all rows or columns | `Calculate average on rows`<br>`Show sum on columns` | Yes |

| Action | Definition of action | Command examples | Needs object/ widget selected? |
|---|---|---|---|
| | | `Aggregate rows as "MyTotal"` | |
| Perform member calculation | Perform calculations on specified members<br>The supported calculations are:<br>• Average<br>• Minimum<br>• Maximum<br>• Median<br>• Aggregate<br>• Rank<br>• Absolute value | `Calculate average for members "Jan", "Feb"`<br>`Create maximum for members "Q1", "Q2"`<br>`Calculate minimum for members "Q1", "Q2"` | Yes |
| Copy | Copy the selected widget | `Copy`<br>`Duplicate` | Yes |
| Swap | Swap rows and columns in a view | `Swap`<br>`Switch`<br>`Flip rows and columns` | Yes |
| Suppress zeroes | Suppress zeroes on the view | `Suppress zeroes on rows`<br>`Hide 0 rows`<br>`Remove cols with value 0` | Yes |
| Display zeroes | Display zeroes previously suppressed | `Display zeroes on rows`<br>`Show 0 rows`<br>`Display zeroes on rows` | Yes |
| Cancel | Cancel the current operation | `Cancel`<br>`Quit` | No |
| Move | Move a dimension/hierarchy to a slot in a view | `Move "Version" to rows`<br>`Transfer "Channel" to bench`<br>`Relocate "Model" to columns` | Yes |
| Refresh | Refresh a view | `Refresh` | Yes |

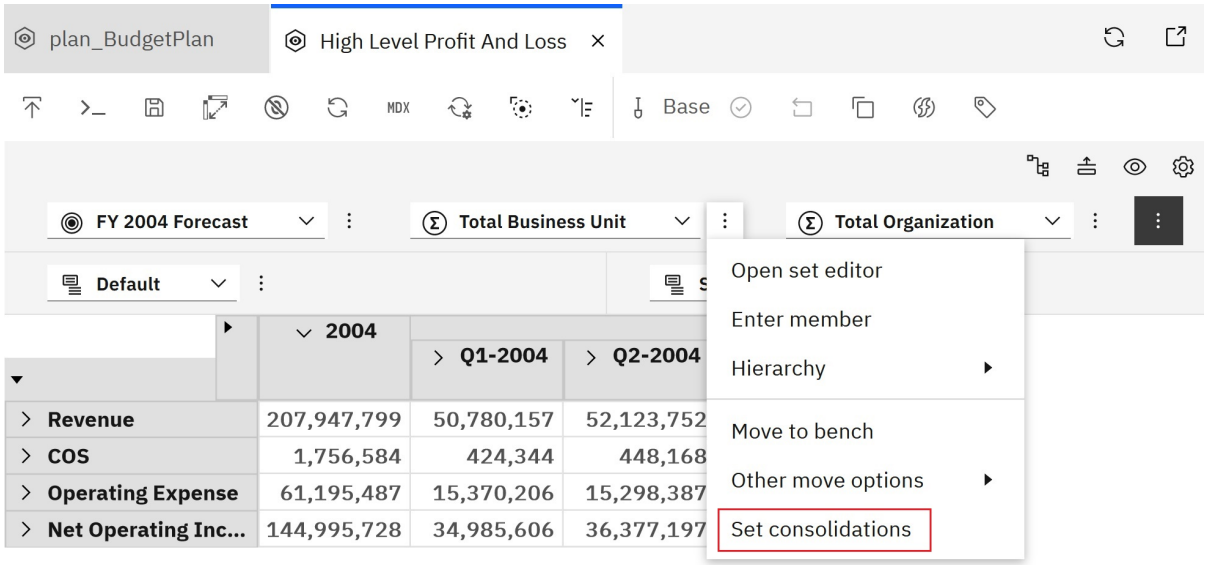
| Action | Definition of action | Command examples | Needs object/ widget selected? |
|---|---|---|---|
| | | `Reload view` | |
| Sort | Sort the rows or columns | `Sort cols by "Month" desc`<br>`Arrange row "Iberia" by column asc`<br>`Sort row "Chile" asc` | Yes |
| Slice | Switch the context of a view to a specific member | `Slice on "Canada"`<br>`Change context to "Budget"` | Yes |
| Create sandbox | Create a sandbox | `Create sandbox "prev_2024"`<br>`Sandbox` | Yes |
| Switch to sandbox | Switch to an existing sandbox | `Switch to "Base" sandbox`<br>`Change sandbox to "MyProjections"` | Yes |
| Expand | Expand a consolidation | `Expand "Total year"`<br>`Drill "Canada"` | Yes |
| Collapse | Collapse a consolidation | `Collapse "Total year"` | Yes |

## Use unnamed members in set consolidations

A set consolidation can now be based on named and unnamed set members.

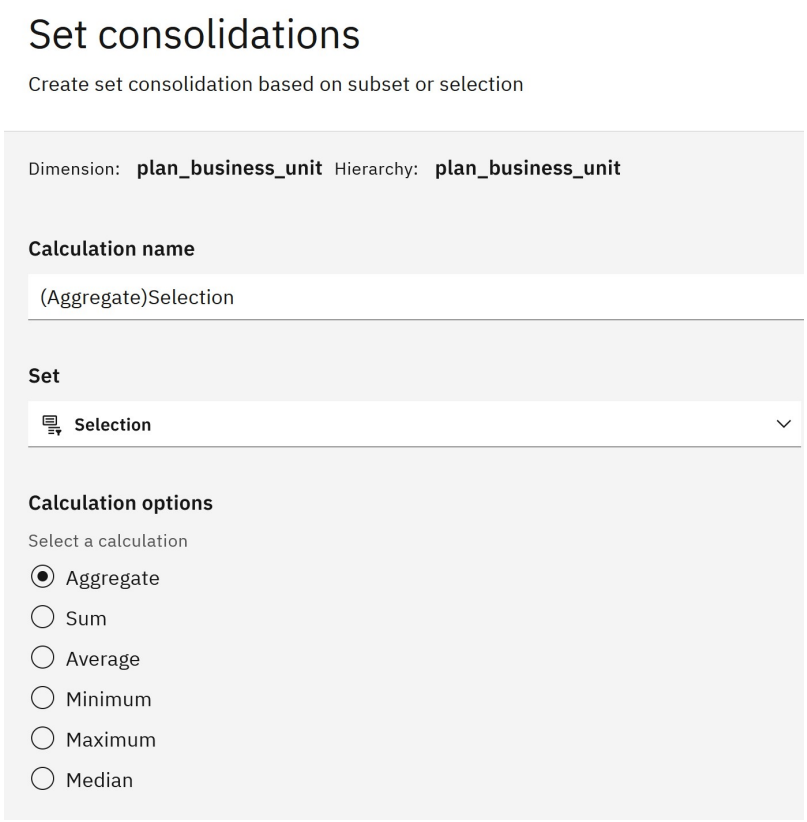
Previously, set consolidations could be based only on existing, named set members.

To create a set consolidation, click the **Set consolidations** menu option. This option is available from multiple places in the application: from the dimension tile overflow menu ⋮ (shown in the image below), from the column and row right-click menu, from the thumbs right-click menu, and also from the **Fields** panel tile menus.

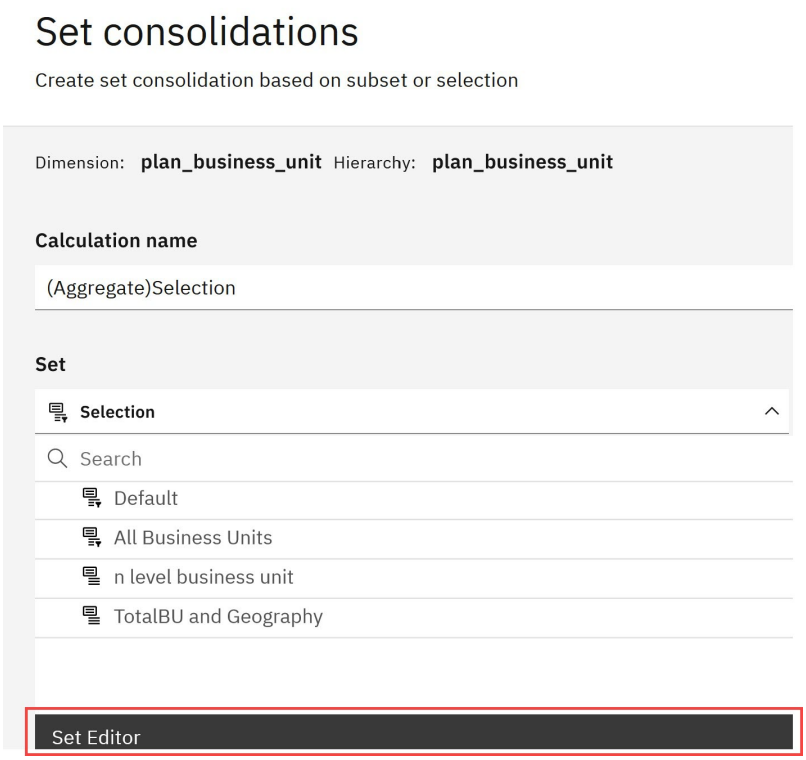



**Tip:** In previous product versions, this option was named **Subset calculations**.

In the **Set consolidations** dialog box, you can select the set (named or unnamed) and calculation type. By default, the **Calculation name** changes based on your selections. If you type a custom name, the name stays constant, no longer reacting to your selections.

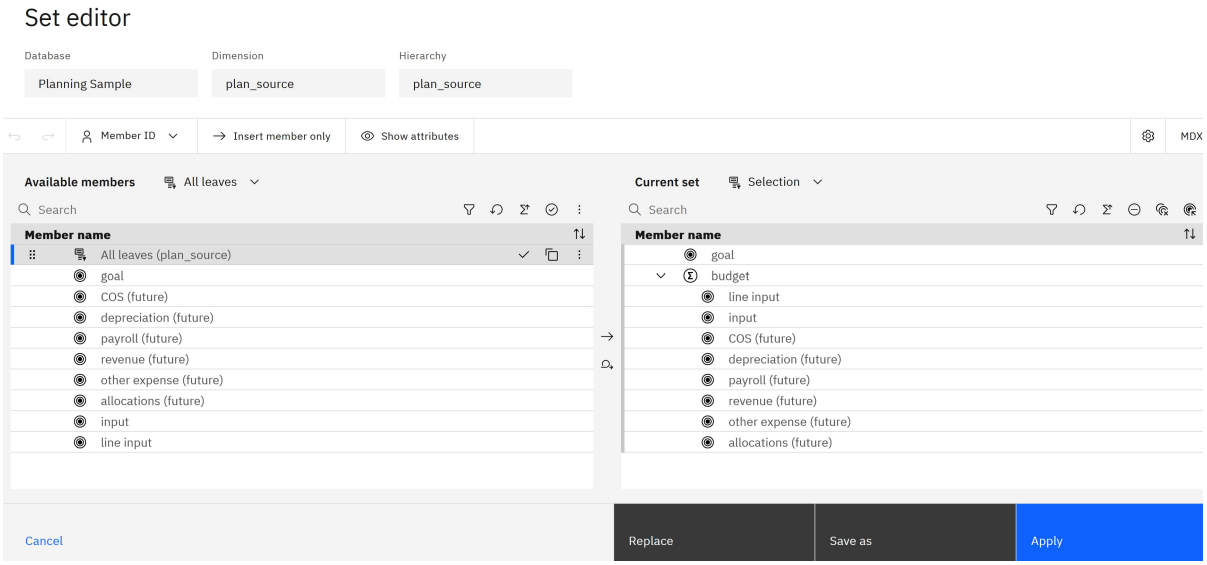



To add an unnamed set member (new in this release), from the **Set** drop-down menu, click **Set Editor**.




In the **Set editor** , select different members to add to the list of sets for your calculation.

**Tip:** Click **Apply** to create an unnamed set with the currently displayed member, or click **Replace** to create an unnamed set with the currently selected members.




The new members are added at the end of the list in the **Set** box, and can be used to create the set consolidation.

For more information, see Add a set consolidation to a view.

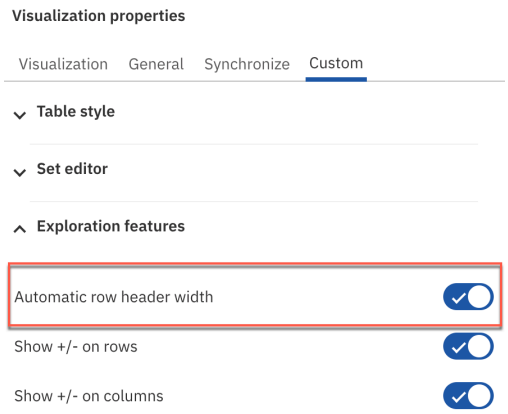
## Lock row header column widths

You can now lock the width of row header columns in Planning Analytics Workspace to prevent the width of row header columns from adjusting automatically.

This prevents the row header columns from changing widths when you expand or collapse consolidation on the rows. The width is locked for any hierarchies or row header columns that are added to the report.

To prevent the row header columns from automatically adjusting their width:

1. Click **Properties**.
2. Navigate to the **Custom** tab.
3. In the **Exploration features** section, enable **Automatic row header width**.



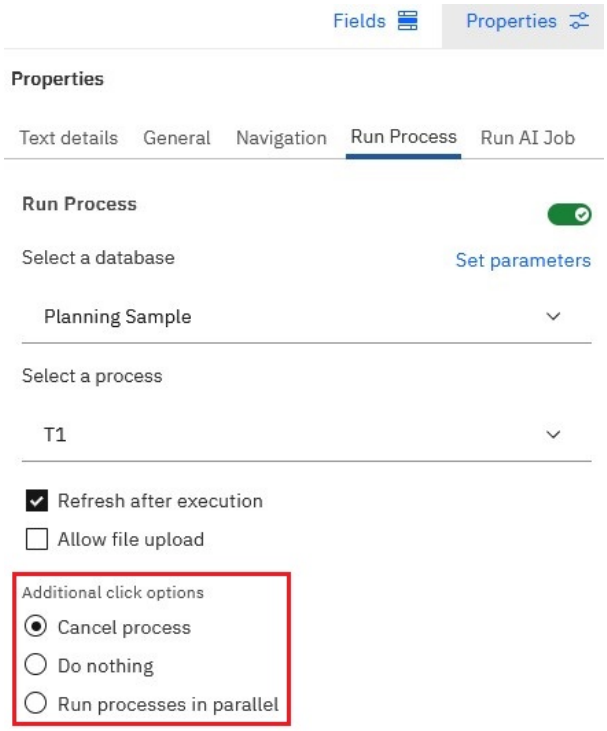

The **Automatic row header width** option is turned on by default.

## Additional click options on an action button

You can configure the behavior of an action button when it is clicked again while already running a TurboIntegrator process.

In previous versions of IBM Planning Analytics Workspace, clicking an action button when it was already running a process canceled the process.

In this version of IBM Planning Analytics Workspace, you can configure the additional click behavior for an action button by using the **Additional click options** settings on the **Run Process** tab on the button **Properties** panel.




The following behavior can be configured for additional button clicks:

**Cancel process**
When an action button that is already running a process is clicked again, the process is canceled, and a related message is displayed. This is the default behavior, which is known from previous versions of the product.

**Do nothing**
When an action button that is already running a process is clicked again the process continues to run. A new process is not started.

**Run processes in parallel**
When an action button that is already running a process is clicked again a new process is started, and the previous process or processes continue to run. This option is useful when the book user wants to run a series of overlapping processes by using the same action button.

You can run a maximum of 5 processes in parallel. If you attempt to run the 6th process, a message about this limit is displayed.

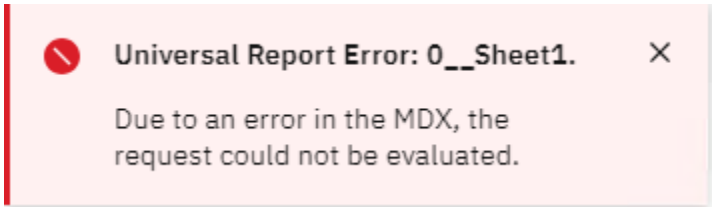
For more information, see Action buttons.

## Improved error messages and error handling

Error toast messages displayed in Universal Report in Planning Analytics Workspace websheets have been improved.

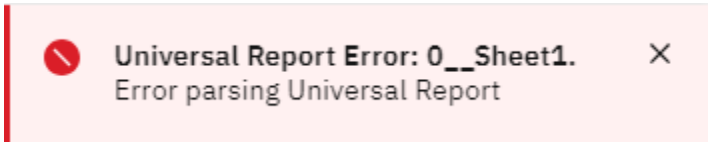
There are two new kinds of error messages that can be displayed in Universal Reports: MDX error messages and parser error messages.

The following image shows the MDX error message:



An MDX result is responsible for the row and column member names, toggles for the row & columns, and the data in their intersections. An MDX query has many elements, including the slicer values, an eligible server name, data source, a cube, and anything else required by your MDX query. When the generated MDX can not be evaluated, an MDX error toast message is shown. If an MDX error is shown, you can click the **Show format areas** button on the IBM Planning Analytics for Microsoft Excel toolbar and check the datasource, the server, the slicers, and the TM1Set formula values for missing or incorrect data. Most of these values are auto-generated when creating a Universal Report from IBM Planning Analytics for Microsoft Excel. More details on the MDX error can be found in the tm1web.log file.

The following image shows the parser error message:




A parser response contains information for the row and column axis, slicers, and hierarchy selections. When one of these key elements cannot be evaluated by the Universal Report, the parser error toast message is shown. If a parser error is shown, you can click the **Show format areas** button on the IBM Planning Analytics for Microsoft Excel toolbar and check the query, row axis sets, and column axis sets for missing or incorrect data. More details on the parser error can be found in the tm1web.log file.

### Show and hide format area

You can now show and hide format areas in Universal Reports and Dynamic Reports in Planning Analytics Workspace Websheets.

There are two new buttons on the toolbar that allow you to show and hide the Universal Reports format area and the Dynamic Reports format area. With the format area shown, you can customize the named ranges.

When you click the **Show sheet format area** icon  , the format area is shown on the websheet.

When you click the **Hide sheet format area** icon  , the format area is hidden on the websheet.

## What's new in modeling - 2.0.96

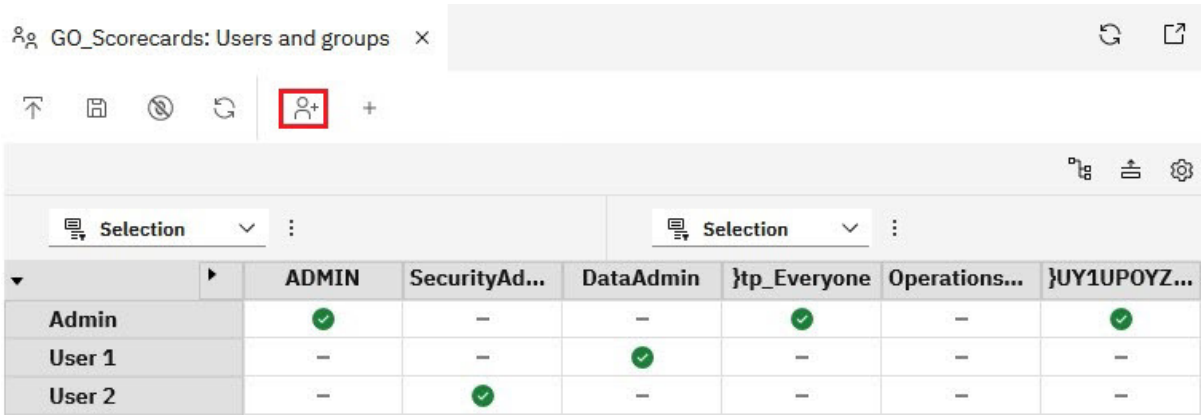
The following new and changed modeling features are available in Planning Analytics Workspace 2.0.96.

### Create users in a Planning Analytics Local database

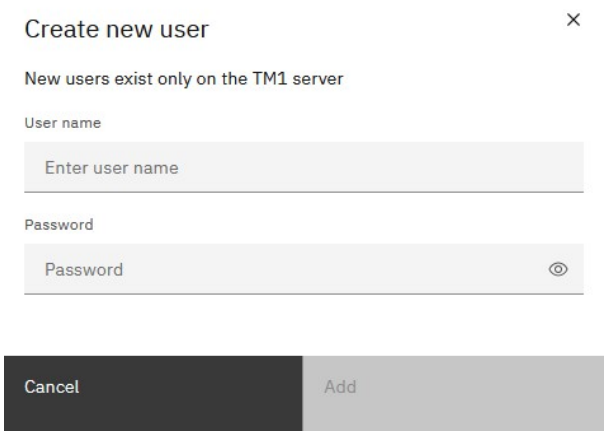
You can now create users in a Planning Analytics Local database using the **Users and groups** editor.

To open the **Users and groups** editor on a modeling workbench, right-click a database in the **Databases** tree, and then click **Manage users and groups**.

The **Users and groups** editor toolbar now includes the **Create new user** icon.

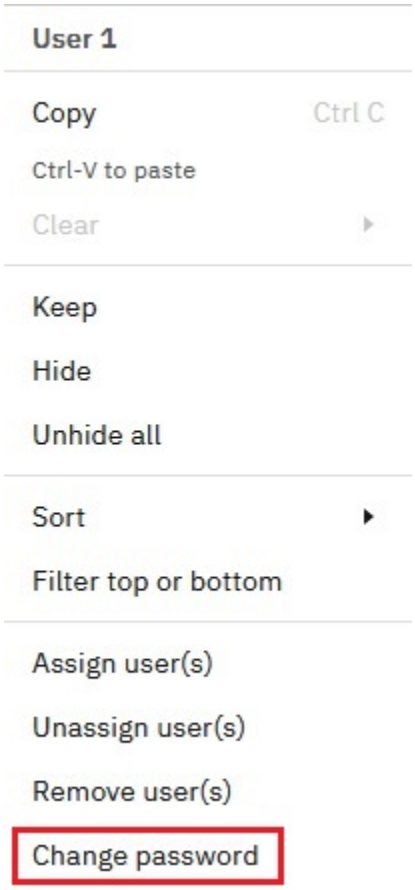



Click this icon to access the **Create new user** dialog box where you type the user name and password. The new user name appears the end of the list of existing users.




The user is added to the Planning Analytics database (TM1 server) only, not to the IBM Planning Analytics Workspace. If you want to use the user for authentication, the user must also exist in the IBM Planning Analytics Workspace.

A related new feature in this release is the ability to modify the user's password. To do that, right-click the user name, and select the **Change password** option from the menu.




For more information, see Create users in a Planning Analytics Local database.

## Localize objects on your database with the Localization editor

A new Localization editor makes it easier to localize the names of cubes, dimensions, members, processes, and chores on your Planning Analytics Workspace database.

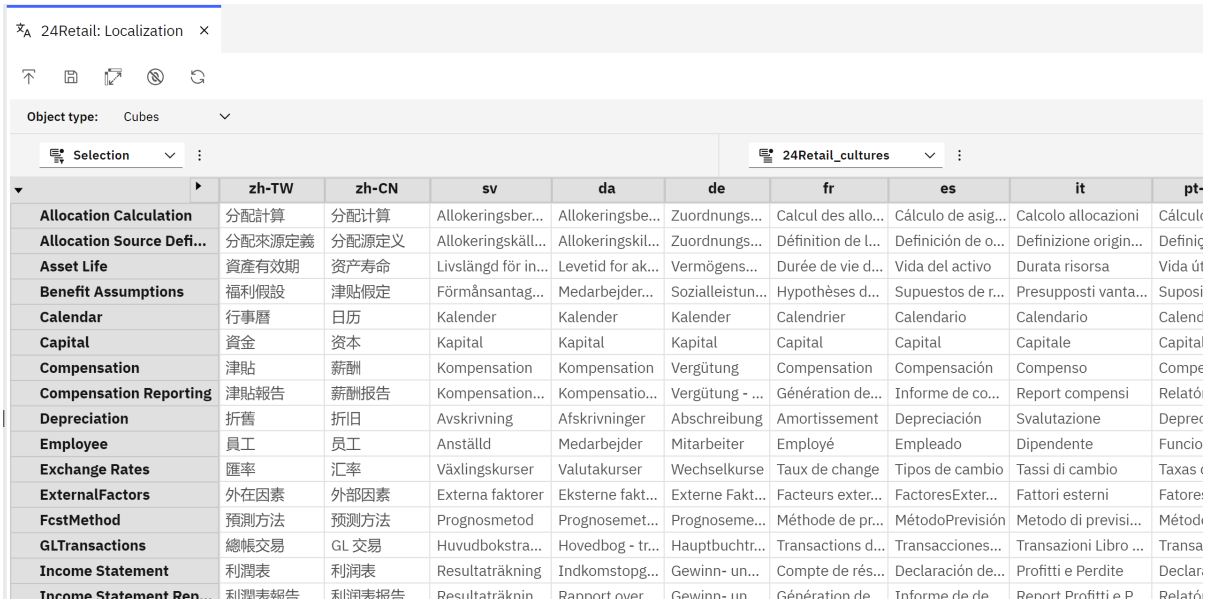
The Localization editor is available from the modeling workbench.

### Localize cubes, dimensions, processes, or chores

To localize cubes, dimensions, processes, or chores, right-click a database on the **Databases** tree, then click **Localization** > **object type**.

You can also open the Localization editor by right-clicking an object node on the Databases tree, then clicking **Localization**. For example, **Cubes** > **Localization** or **Chores** > **Localization**.

The Localization editor displays a grid, with object names on the rows and the languages included in the }Cultures control dimension on the columns.




| | zh-TW | zh-CN | sv | da | de | fr | es | it | pt- |
|---|---|---|---|---|---|---|---|---|---|
| Allocation Calculation | 分配計算 | 分配计算 | Allokeringsber... | Allokeringsbe... | Zuordnungs... | Calcul des allo... | Cálculo de asig... | Calcolo allocazioni | Cálcul |
| Allocation Source Defi... | 分配來源定義 | 分配源定义 | Allokeringskäll... | Allokeringskil... | Zuordnungs... | Définition de l... | Definición de o... | Definizione origin... | Definiç |
| Asset Life | 資產有效期 | 资产寿命 | Livslängd för in... | Levetid for ak... | Vermögens... | Durée de vie d... | Vida del activo | Durata risorsa | Vida út |
| Benefit Assumptions | 福利假設 | 津贴假定 | Förmånsantag... | Medarbejder... | Sozialleistun... | Hypothèses d... | Supuestos de r... | Presupposti vanta... | Suposi |
| Calendar | 行事曆 | 日历 | Kalender | Kalender | Kalender | Calendrier | Calendario | Calendario | Calend |
| Capital | 資金 | 资本 | Kapital | Kapital | Kapital | Capital | Capital | Capitale | Capital |
| Compensation | 津貼 | 薪酬 | Kompensation | Kompensation | Vergütung | Compensation | Compensación | Compenso | Compe |
| Compensation Reporting | 津貼報告 | 薪酬报告 | Kompensation... | Kompensatio... | Vergütung - ... | Génération de... | Informe de co... | Report compensi | Relató |
| Depreciation | 折舊 | 折旧 | Avskrivning | Afskrivninger | Abschreibung | Amortissement | Depreciación | Svalutazione | Deprec |
| Employee | 員工 | 员工 | Anställd | Medarbejder | Mitarbeiter | Employé | Empleado | Dipendente | Funcio |
| Exchange Rates | 匯率 | 汇率 | Växlingskurser | Valutakurser | Wechselkurse | Taux de change | Tipos de cambio | Tassi di cambio | Taxas |
| ExternalFactors | 外在因素 | 外部因素 | Externa faktorer | Eksterne fakt... | Externe Fakt... | Facteurs exter... | FactoresExter... | Fattori esterni | Fatores |
| FcstMethod | 預測方法 | 预测方法 | Prognosmetod | Prognosemet... | Prognoseme... | Méthode de pr... | MétodoPrevisión | Metodo di previsi... | Métod |
| GLTransactions | 總帳交易 | GL 交易 | Huvudbokstra... | Hovedbog - tr... | Hauptbuchtr... | Transactions d... | Transacciones... | Transazioni Libro ... | Transa |
| Income Statement | 利潤表 | 利润表 | Resultaträkning | Indkomstopg... | Gewinn- un... | Compte de rés... | Declaración de... | Profitti e Perdite | Declar |
| Income Statement Rep... | 利潤表報告 | 利润表报告 | Resultaträknin... | Rapport over... | Gewinn- un... | Génération de... | Informe de de... | Report Profitti e P... | Relató |

You can click the **Object type** menu and select a different object type to localize.

The first time you open the Localization editor for an object type, a new localization control cube is created on the database where the objects reside. Localized object names are saved in these control cubes.

| Object type | Localization control cube |
|---|---|
| Cubes | }LocalizedCubeAttributes |
| Dimension | }LocalizedDimensionAttributes |
| Processes | }LocalizedProcessAttributes |
| Chores | }LocalizedChoresAttributes |

To localize object names, you can work directly in the Localization editor or upload data to the appropriate localization control cube.

**Localize object names in the Localization editor**
Click the cell at the intersection of an object name and a language, then enter the appropriate localized name.

**Upload data to a localization control cube**
Right-click the appropriate localization control cube on the **Databases** tree, then click **Import data**. See Importing data into a cube for more information.

## Localize dimension members

To localize dimension member names, right-click an individual dimension on the **Databases** tree, then click **Localization**.

The Localization editor displays a grid, with member names on the rows and the languages included in the }Cultures control dimension on the columns.

The first time you open the Localization editor for members in a specific dimension, a new localization control cube is created on the database where the dimension resides. The localization control cube is named `}LocalizedElementAttributes_`*`dimName`*. Localized dimension member names are saved in this control cube.

For example, when you first open the Region dimension in the Localization editor, the `}LocalizedElementAttributes_Region` control cube is created.

To localize dimension member names, you can work directly in the Localization editor or upload data to the appropriate localization control cube.

**Localize member names in the Localization editor**
Click the cell at the intersection of a dimension member name and a language, then enter the appropriate localized name.

**Upload data to a localization control cube**
Right-click the appropriate }LocalizedElementAttributes_*dimName* localization control cube on the **Databases** tree, then click **Import data**. See Importing data into a cube for more information.

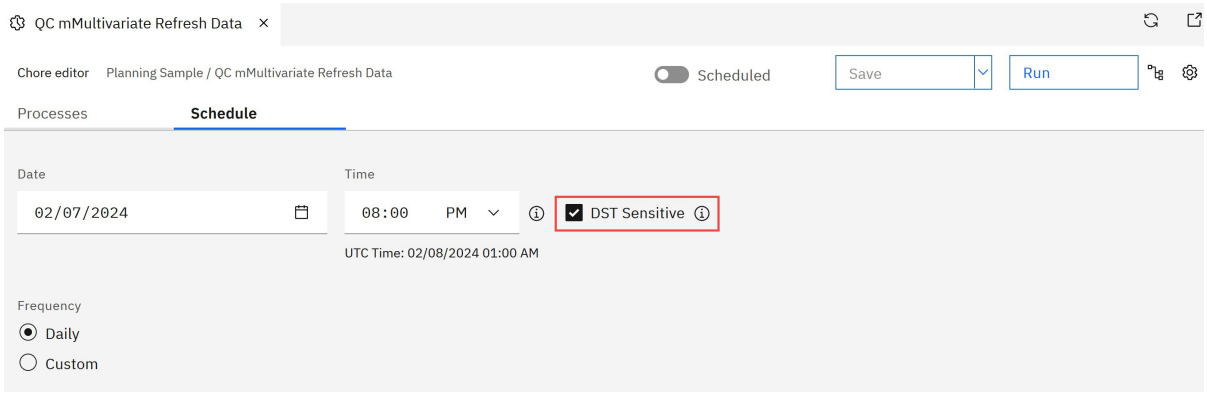
## Use improved auto-complete and validation to simplify rules creation

An improved auto-complete feature better identifies the required components of a rule statement and presents the relevant components for selection when you press CTRL+SPACE in the rules editor.

Additionally, you can now click the **Validate business rules** icon to fully validate your rules. This performs a validation that is equivalent to that which is performed when you save a rule and exceeds the real-time syntax checking and validation that occurs while creating a rule.

## Schedule chores using Daylight Saving Time (DST)

The chore editor now includes the **DST Sensitive** option on the **Schedule** tab.




When the **DST Sensitive** checkbox is selected (default), the chore time is automatically updated when Daylight Saving Time (DST) starts or ends for the region in which the Planning Analytics database is located.

For more information, see Create and edit chores.

## Preview a location data source in the process editor

Prior to Planning Analytics Workspace 2.0.96, a preview of a location data source could not be examined while creating or editing a process in the process editor.

You can now preview a location source in the process editor. Click **Preview** after you specify the location to review the structure of the data source.

For more information about using a location data source, see Define a location data source.

# What's new in administration - 2.0.96

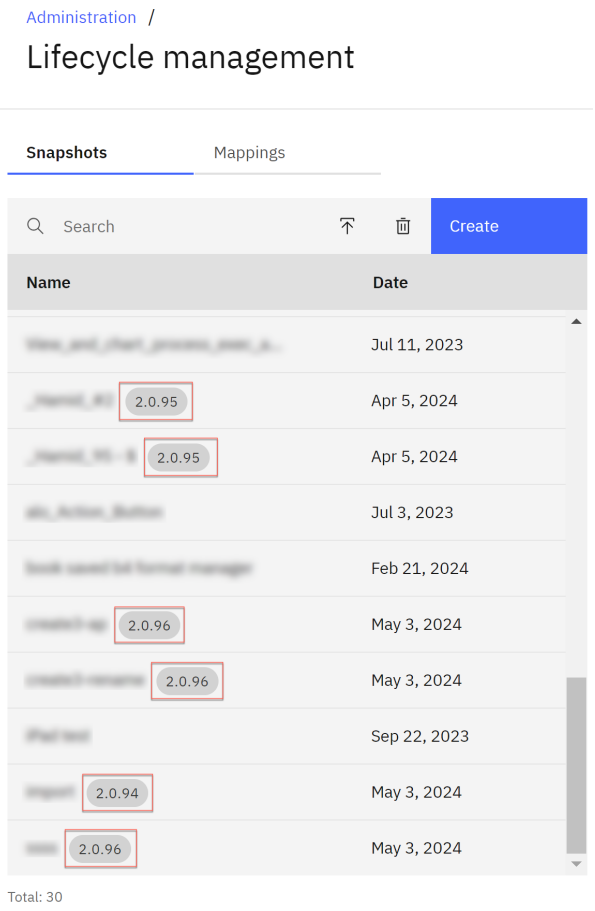
The following new and changed administration features are available in Planning Analytics Workspace 2.0.96.

## View the version of IBM Planning Analytics Workspace for a snapshot

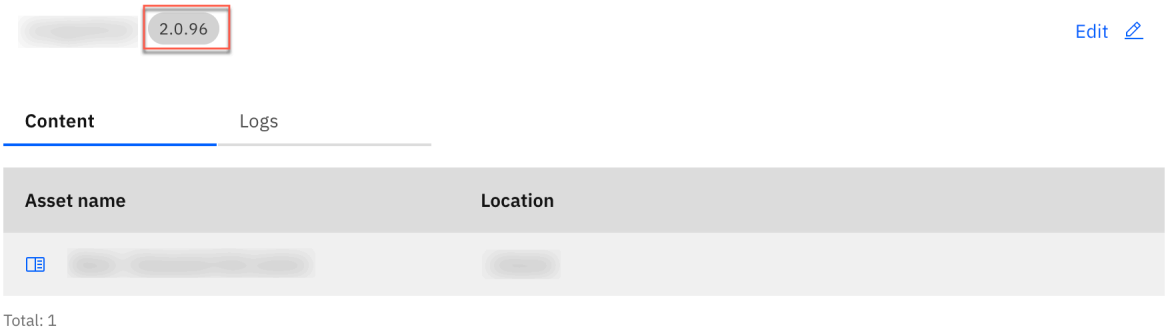
In IBM Planning Analytics Workspace Administration, you can see the version of Planning Analytics Workspace that a snapshot was created in.

This feature allows you to identify which snapshots are older.

The version of Planning Analytics Workspace appears beside the snapshot name in the snapshot list.




The version number also appears beside the snapshot name on the **Details** panel.




**Note:** This feature is only available in IBM Planning Analytics Workspace 2.0.96 or later. A snapshot can display versions 2.0.90 or later. If the snapshot version is not displaying, you may need to export the snapshot and import it again.

For more information on snapshots, see Create a snapshot.

## Deprecation notices - 2.0.96

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.96.

### End of support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 (Local only)

Support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 will end in June 2024.

This coincides with Red Hat's announced end of maintenance support for Red Hat Enterprise Linux 7. For more information, see https://www.redhat.com/en/blog/end-maintenance-red-hat-enterprise-linux-7-almost-here.

Planning Analytics Workspace Local versions 2.0.70 and later are supported on Red Hat Enterprise Linux version 8.

Planning Analytics Workspace Local versions 2.0.93 and later are supported on Red Hat Enterprise Linux version 9.

### End of support for Planning Analytics Workspace Local on Windows Server 2016 (Local only)

Windows Server 2016 is no longer supported on Planning Analytics Workspace Local versions 2.0.96 and later.

This is in accordance with Microsoft's announced end of Mainstream Support for Windows Server 2016 as of January 11, 2022. For more information, see https://learn.microsoft.com/en-us/lifecycle/products/windows-server-2016.

Planning Analytics Workspace Local versions 2.0.74 and later are supported on Windows Server 2019.

You can continue to use Windows Server 2016 with Planning Analytics Workspace Local versions 2.0.95 and prior.

# 2.0.95 - What's new, April 23, 2024

Learn about new features and known issues in version 2.0.95 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

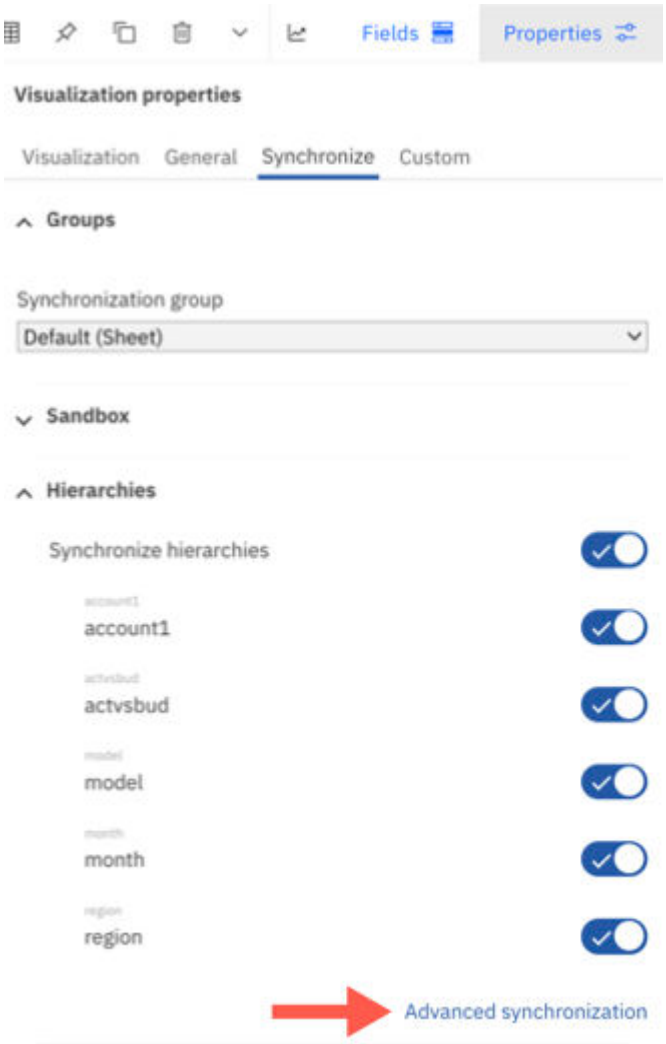
## What's new in books and reporting - 2.0.95

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.95.

## New synchronization settings allow customized display of hierarchies

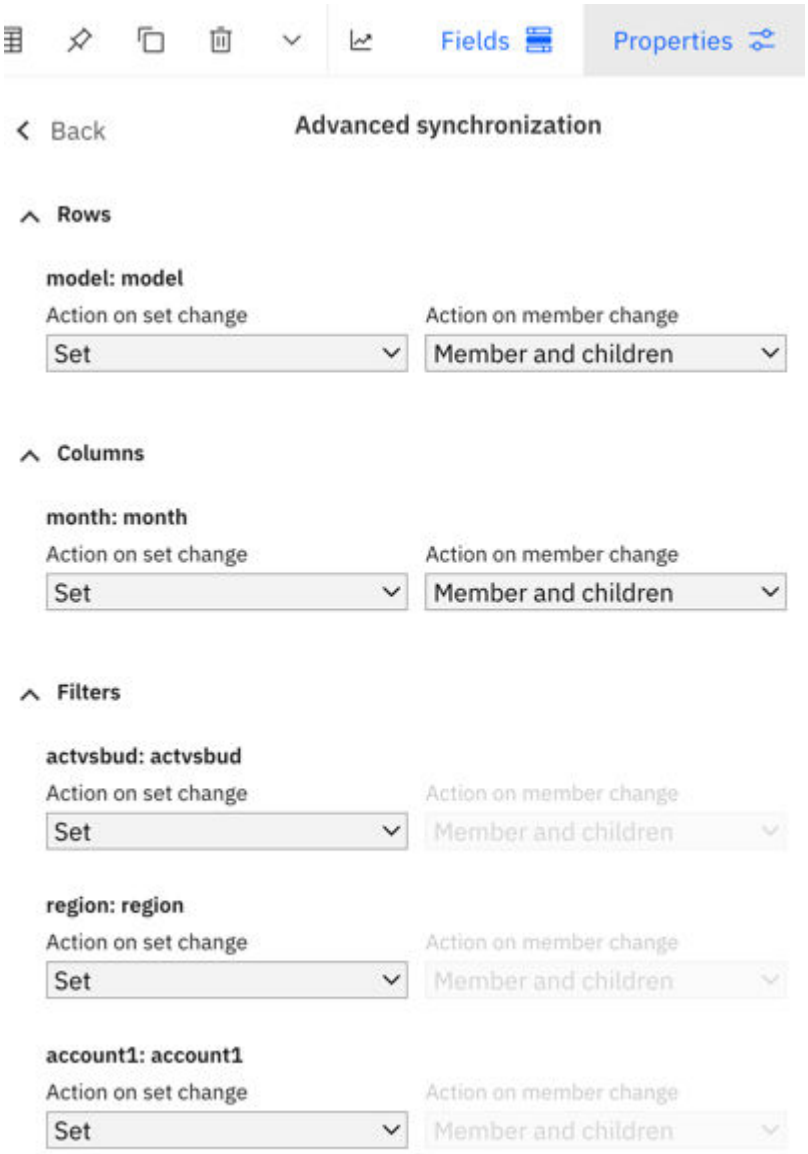
New synchronization settings under **Advanced synchronization** now let you customize how each hierarchy in the cube viewer or visualization displays when a synchronization event is received by that widget.

You can find **Advanced synchronization** in the **Properties > Synchronize > Hierarchies** section when synchronization is enabled for the widget that receives the synchronization.




**Note:** The advanced synchronization settings apply to only the widget that receives the synchronization event.

The **Advanced synchronization** panel is divided into **Row**, **Column**, and **Filters** sections. You can choose what each hierarchy on the **Row** and **Column** areas displays when the widget receives a set change or member change synchronization event. You can select the hierarchy display for the **Context** area under **Filters**.




For a set change synchronization event, you can choose what you want the set in the receiving widget to display:

- Set - The set with all its members and children displays in the widget.
- Set and aggregate - The aggregate of the all the members in the set shows below the set.
- Aggregate and set - The aggregate of the all the members in the set shows above the set.
- Aggregate - Only the aggregate of all the members in the set displays in the widget.

Similarly, for a member synchronization event, you can choose what you want the set in the receiving widget to display:

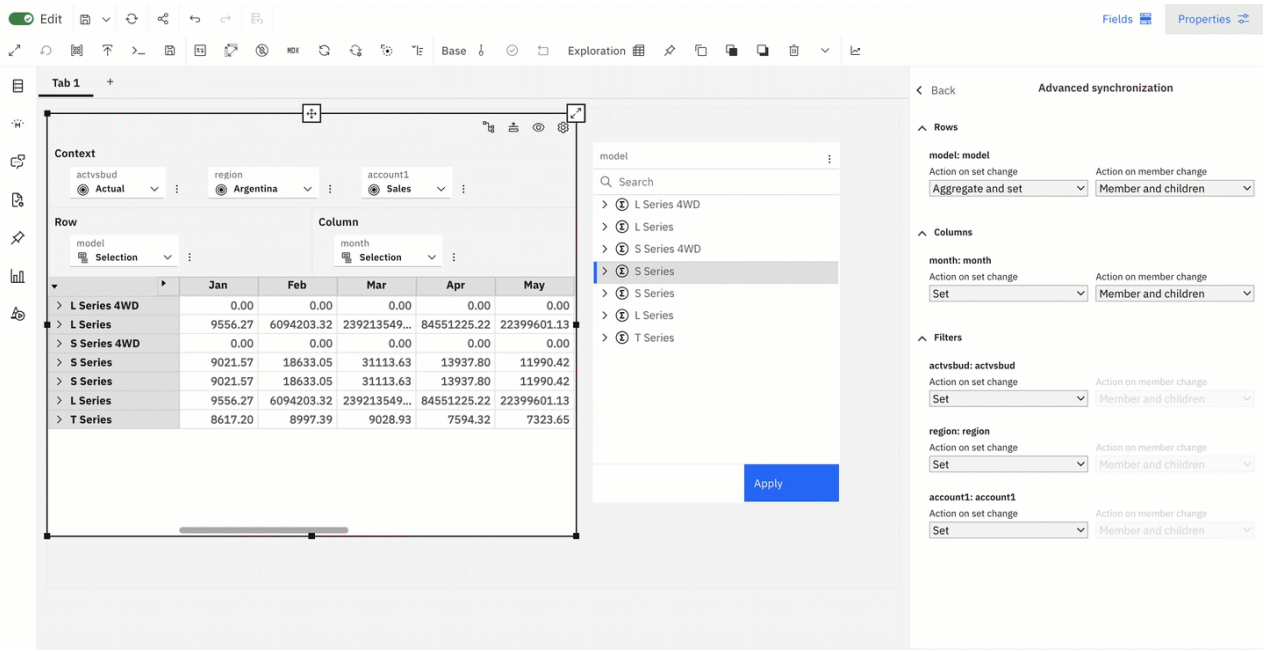
- Member and children
- Member
- Children
- Member and descendants
- Descendants
- Member and parents

- Parents
- Member and ancestors
- Ancestors

The advanced synchronization settings persist if you move a hierarchy from to a different area in the widget.

If you're using an advanced selector widget to synchronize members, you can select multiple members and the receiving widget will display the set and members based on the synchronization settings you selected for that widget.

In the following example, advanced synchronization is set to **Aggregate and set** and **Member and children** for the *model* hierarchy on the **Row**. An advanced selector list for *model* is used to synchronize members. When the member is changed to *L Series*, the cubeviewer updates to display an aggregate of all the members in the set and the set updates to display only the selected member and its children.



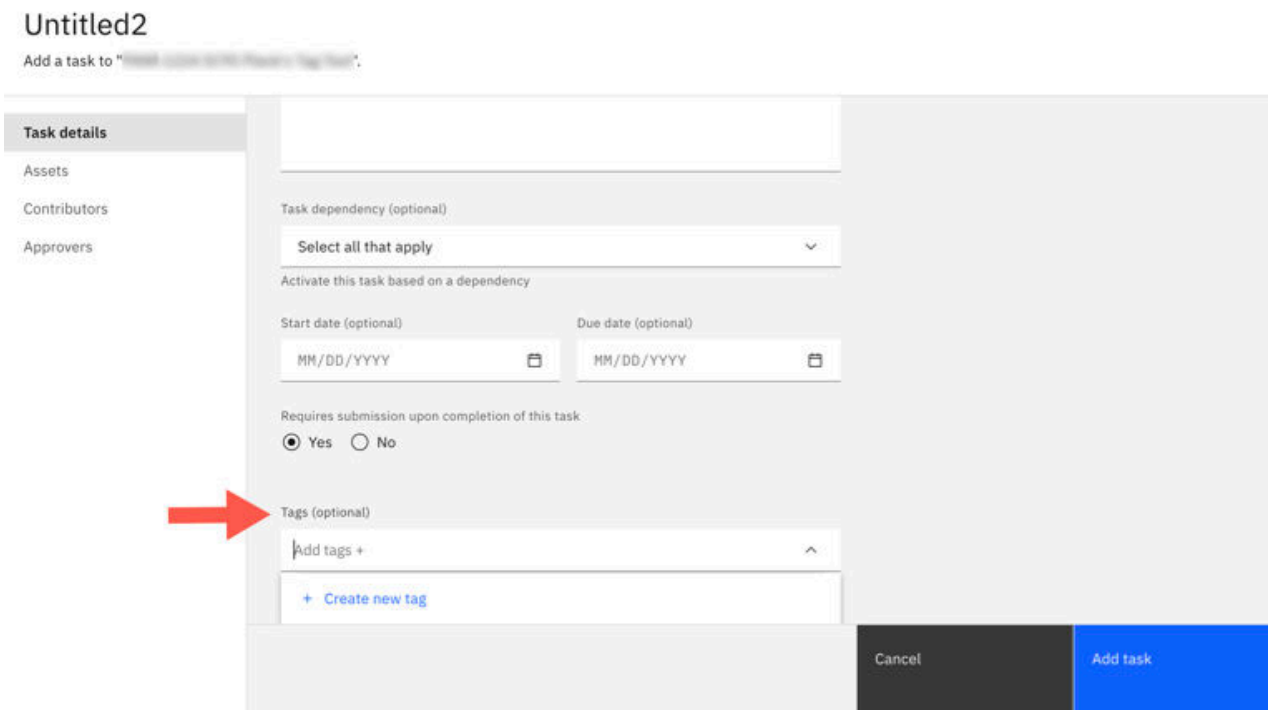

# What's new in plans and applications - 2.0.95

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.95.

## Create color-coded tags in plan tasks

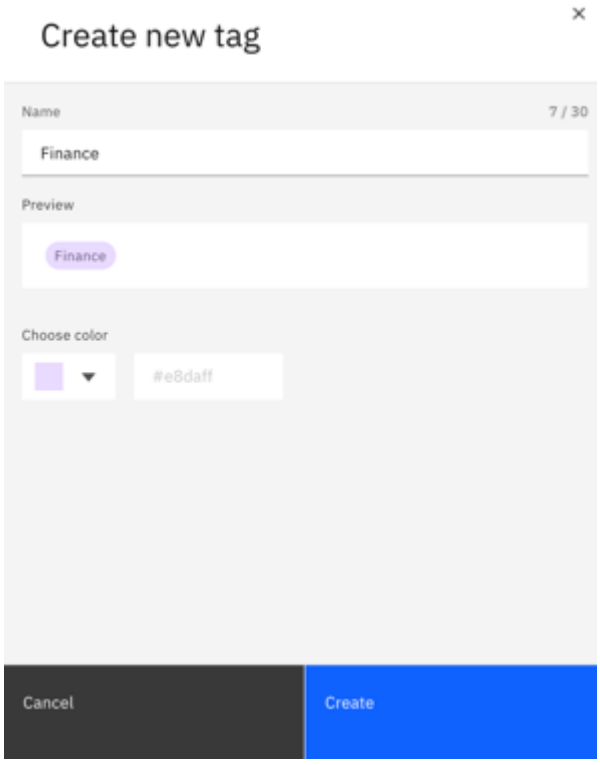
You can create color-coded, custom tags in plan tasks to quickly identify tasks on the plan Overview page.

You can create tags when creating or editing a task.

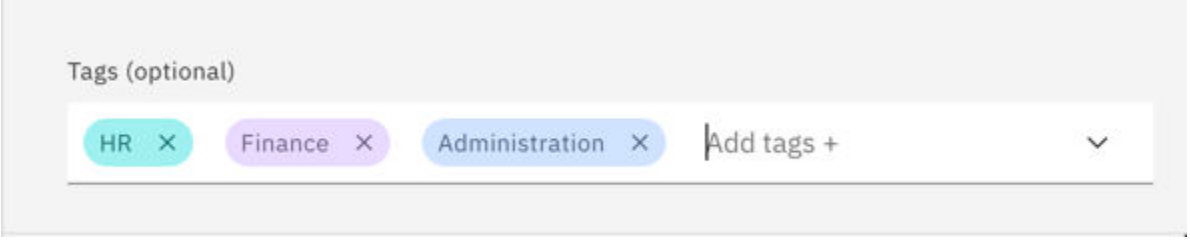



To create and add a new tag to your task, follow these steps:

1. On the Task details tab in a plan task, click into the **Tags** field and click **Create new tag**.
2. In the **Name** field, enter a label for your tag.
3. Choose a color for your tag. You can see what the tag will look like in the **Preview** field.




4. Click **Create**.
5. Expand the **Tags** field, select the tag that you just created to add it to the task.




Any tags that you create are saved to the plan. This means that all tasks in a plan can share the available tags in that same plan. Tasks can't access tags from other plans.

Click the **Edit a tag** icon next to a tag to edit or delete the tag.

**Important:** When you delete a tag, the tag is removed from all tasks in the plan that use the tag.

# What's new in modeling - 2.0.95

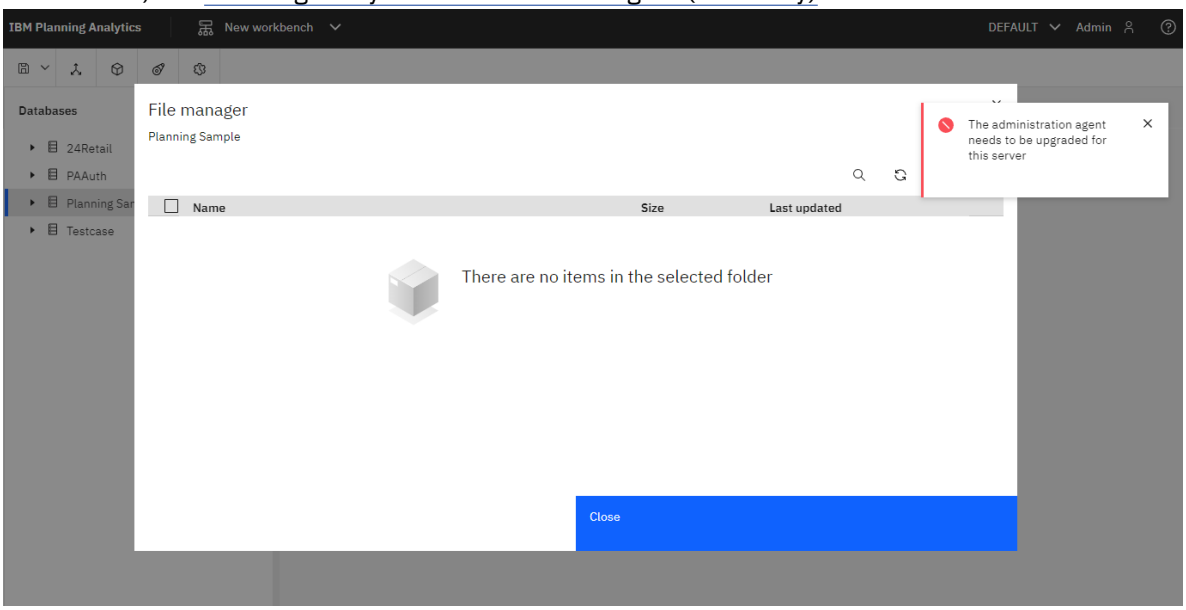
The following new and changed modeling features are available in Planning Analytics Workspace 2.0.95.

## Enhancements to the File manager

The **File manager** tool on the modeling workbench features several enhancements in this release that lets you easily move files around. The **File manager** shows a list of all the files and folders that have been uploaded to the Planning Analytics database and allows you to manage those items.

The **File manager** enhancements described here are available only in Planning Analytics Workspace on Cloud and Planning Analytics Workspace Local. They are not available in Planning Analytics Workspace on Planning Analytics as a Service or when connected to TM1 Database 12.

**Attention:** If you're using IBM Planning Analytics Workspace Local, update the Planning Analytics Administration agent to the version that is included in Planning Analytics Workspace 2.0.95. You won't be able to use the **File manager** in Planning Analytics Workspace 2.0.95 unless you update the Planning Analytics Administration agent to the version from the same release. For more information, see Planning Analytics Administration agent (local only).




The File manager in Planning Analytics Workspace now supports the following actions:

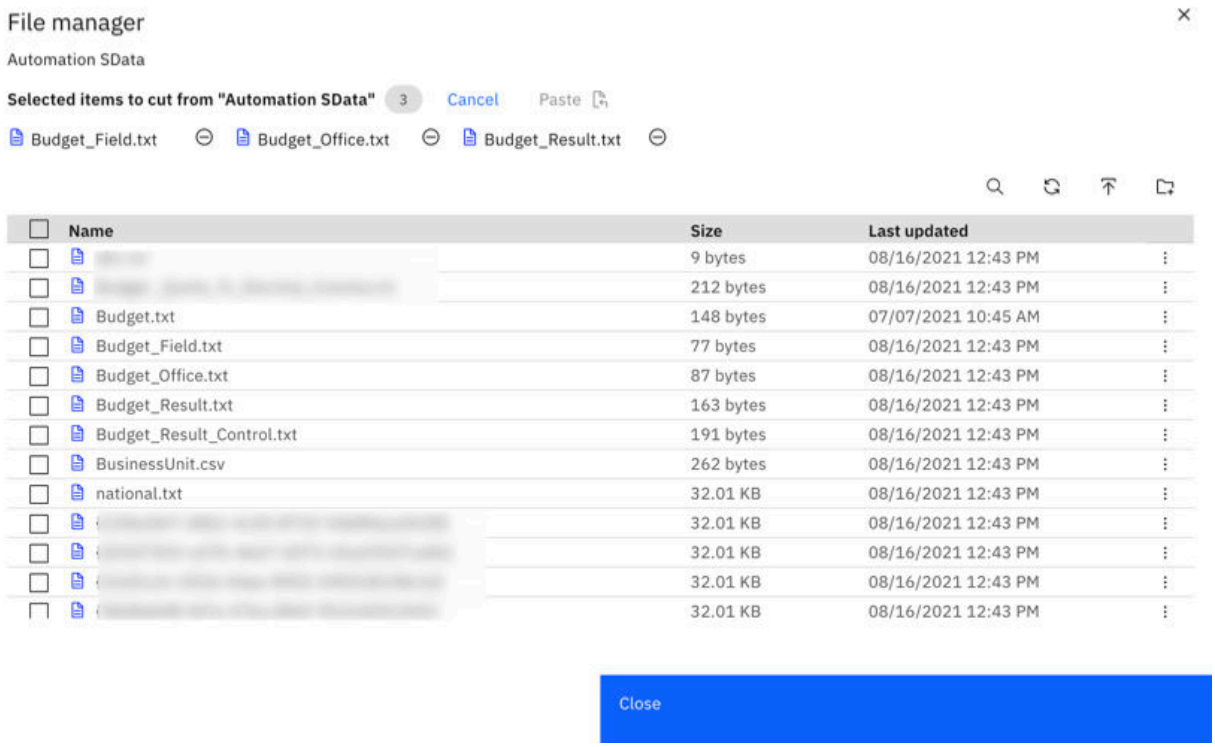
**Create folders and subfolder**

You can now create folders and subfolders in the **File manager**. If you delete a folder, subfolders or files in that folder are also deleted.

**Cut, copy, and paste files**

You can now move or copy files or folders to a different folder within the **File manager**. The **Cut** and **Copy** options display in the toolbar when you select one or more items.

To cut or copy, select one or more items and click **Cut** or **Copy**. Your selections display in alphabetical order as a horizontal list for a quick review before you cut or copy the items.




After you review the list of items that you want to cut or copy, you can paste the items to a different folder. To paste a cut or copied item, go to the destination folder and click **Paste**.

**Compress and expand files**

Select one or more files or folders and click **Compress** to compress the items into a .zip file. You can compress a file and move it to a new location without using **Cut**. To do this, select a file and click **Compress**. Then, go to the destination folder and click **Compress** again.

To expand a compressed file, select the file and click **Expand**. As with compressing a file to a new location, you can expand a file to a new folder.

## Roll back transactions in the transaction log

You can now roll back one or more transactions from the database transaction log in Planning Analytics Workspace.

### About this task

### Procedure

1. Open the transaction log from the **Databases** tree on a modeling workbench.
   a) To open the transaction log at the database level, right-click a database, then click **Logs** > **Transaction log**.
   b) To open the transaction log for an individual cube, right-click the cube, then click **Logs** > **Transaction log**.
2. Set filtering options if desired, then click **Search**. You can also click any column heading to sort transactions.

3. Select the transactions you want to roll back. You can roll back up to 1500 transactions in a single roll back action.
4. Click **Roll back** to roll back a single selected transaction or click **Roll back all** to roll back multiple selected transactions..
5. Click **OK** to confirm the action.

   The current cell value is reverted to the value indicated by the **Prior value** column. If multiple selections target the same cell, the most recent transaction takes precedence.

## Manage database thread activity on the modeling workbench

You can now view and manage thread activity for a database directly on the modeling workbench.

This new feature provides many of the same capabilities available for thread monitoring in Planning Analytics Administration.

To monitor thread activity on a Planning Analytics database, right-click the database on the **Databases** tree, then click **Threads**.

The thread monitor displays all threads on the database using these sortable columns. The availability of columns varies between Planning Analytics database and TM1 Database 12. You can click **Settings**  to manage the display of columns.

**Name**
The name of the user who initiated the thread.

**State**
The current thread state.

- Idle: not using resources
- Wait: waiting on other threads, blocked
- Running
- Commit: committing updates made by a thread
- Rollback: rolling back to a state before an update
- Login: when a user logs in and initiates a thread

**Type**
Indicates a User or System thread.

**Function**
The API function being executed by the thread.

**Wait**
The length of time, in seconds, that the thread has been in a wait state.

**Elapsed**
The total elapsed time of the thread in its current state.

**W/R/Ix locks**
The number of write, read, and intent-to-write locks that the thread holds.

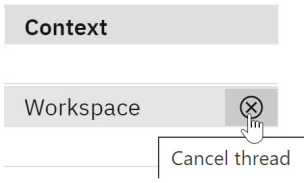
A write lock allows only the thread that holds the lock to access and write changes to an object. No other thread can read or modify the object until the W lock has been released.

A read lock allows many threads to read from an object at the same time, but does not allow another thread to modify or write to the object until all R locks have been released.

An intent-to-write lock reserves the right for a thread to obtain a write lock on an object when all read locks have been released. Only one thread at a time is allowed to have an Ix lock on an object.

**Context**
The connected Planning Analytics client that is associated with the thread.

To cancel an individual thread, click the **Cancel** icon adjacent to the final column on the report, then confirm the action.




To cancel multiple threads:

1. Click **Pause**. When monitoring is paused, check boxes are available to select individual threads.
2. Select the threads you want to cancel.
3. Click **Cancel threads** on the toolbar, then confirm the action.
4. Click **Resume** to resume thread monitoring.

## Deprecation notices - 2.0.95

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.95.

### End of support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 (Local only)

Support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 will end in June 2024.

This coincides with Red Hat's announced end of maintenance support for Red Hat Enterprise Linux 7. For more information, see https://www.redhat.com/en/blog/end-maintenance-red-hat-enterprise-linux-7-almost-here.

Planning Analytics Workspace Local versions 2.0.70 and later are supported on Red Hat Enterprise Linux version 8.

Planning Analytics Workspace Local versions 2.0.93 and later are supported on Red Hat Enterprise Linux version 9.

### End of support for Planning Analytics Workspace Local on Windows Server 2016 (Local only)

Support for Planning Analytics Workspace Local on Windows Server 2016 will end in 2024.

This is in accordance with Microsoft's announced end of Mainstream Support for Windows Server 2016 as of January 11, 2022. For more information, see https://learn.microsoft.com/en-us/lifecycle/products/windows-server-2016.

Planning Analytics Workspace Local versions 2.0.74 and later are supported on Windows Server 2019.

# 2.0.94 - What's new, March 20, 2024

Learn about new features and known issues in version 2.0.94 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative.

For a full accounting of new and changed features by component, see the Planning Analytics New Features Guide.

.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

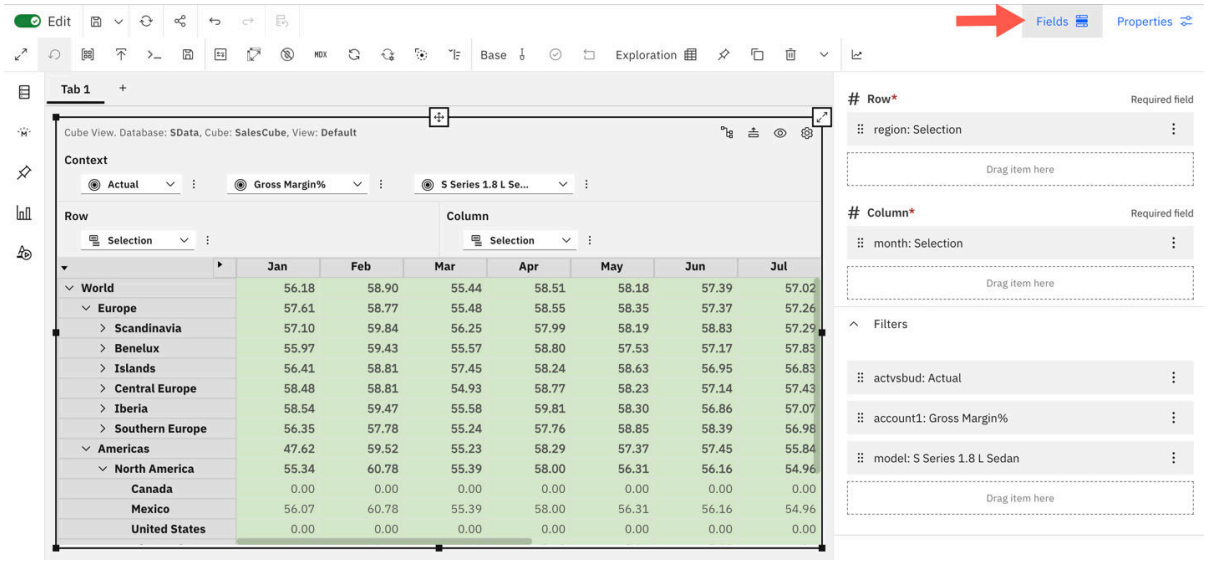
You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

# What's new in books and reporting - 2.0.94

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.94.

## Fields panel now available for explorations

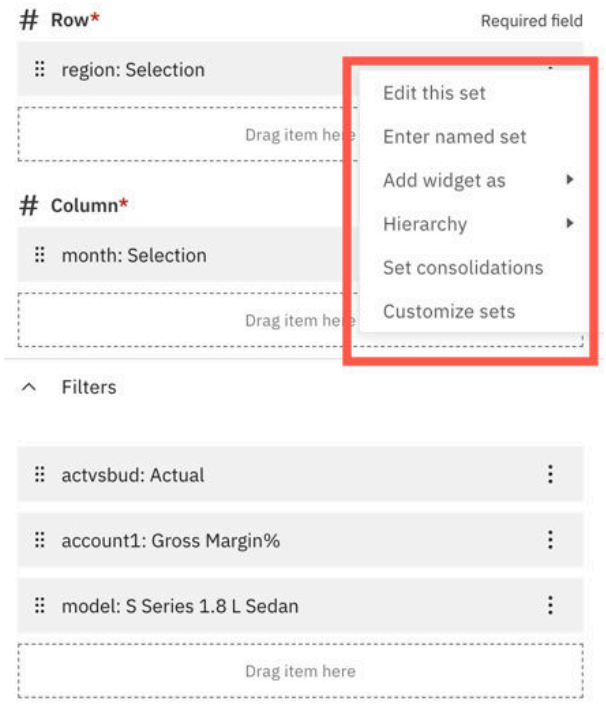
The **Fields** panel is now available for explorations in books. Previously, you could not use the **Fields** panel to manage settings for explorations.




All of the actions that are available in the Overview area of an exploration are supported in the **Fields** panel, except **Move to bench**.

Select an exploration and go to **Fields** to open the fields settings. Dimensions that are in the **Context** area of the exploration appear under **Filters** in the **Fields** panel. You can drag dimensions from **Filters** to the **Row** and **Column** fields.

Click the **More** icon next to a dimension in the **Row** or **Column** field to edit the set, remove the hierarchy, create a subset calculation (set consolidations), and customize sets. You can also add a Selector list, Selector tile, or an Advanced selector widget to the book from this menu.

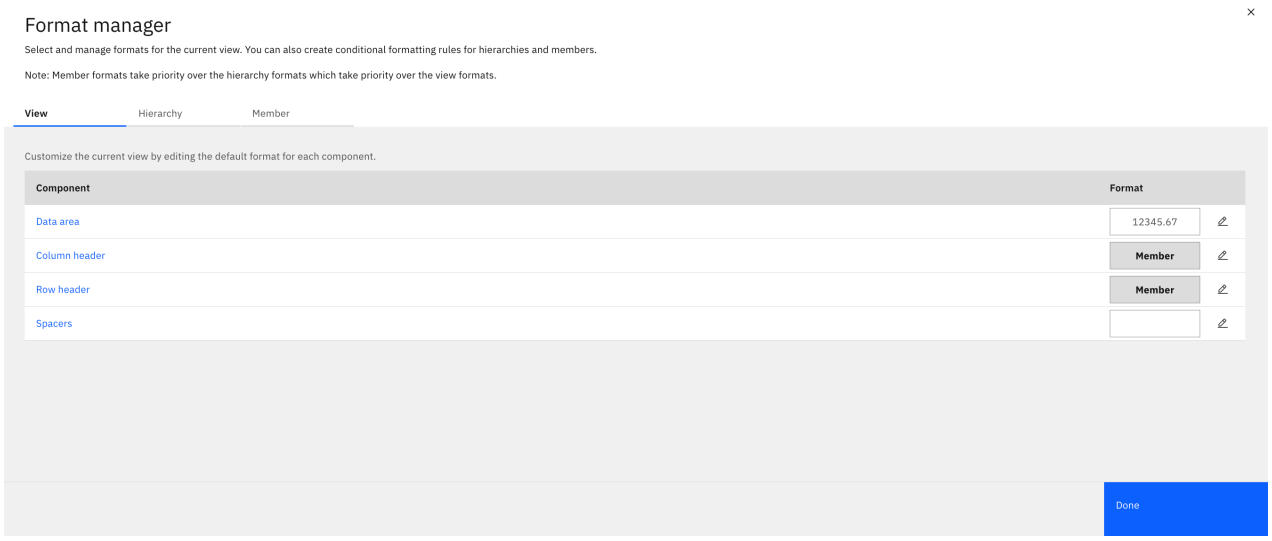



## Use the new Format manager to format cube views

All formatting options for the cube viewer are now consolidated into the new **Format manager**.

Formatting options that were previously under **Format values** in the toolbar, cube view menus (Formats, Cells alignment, Member alignment, and Conditional rules), and **Visualization properties** (Data cells, Row header, and Column header sections) can now all be accessed in the **Format manager**.

**Important:** The new **Format manager** is subject to enablement by a Planning Analytics Workspace administrator. If an administrator does not explicitly enable the new **Format manager** feature, all cube viewer formatting options remain in their existing locations until the enablement release for this feature.




You can apply and manage the following formats for a view and members in the current view:

### View

- Cell color for data cells, row headers, and column headers
- Data formats for data cells

- Font and string formats for data cells, row headers, and column headers

You can easily apply common formats in the view with the **Inherit** toggle. When you set formats in a view, you can inherit formatting from data cells and apply it to row or column headers.

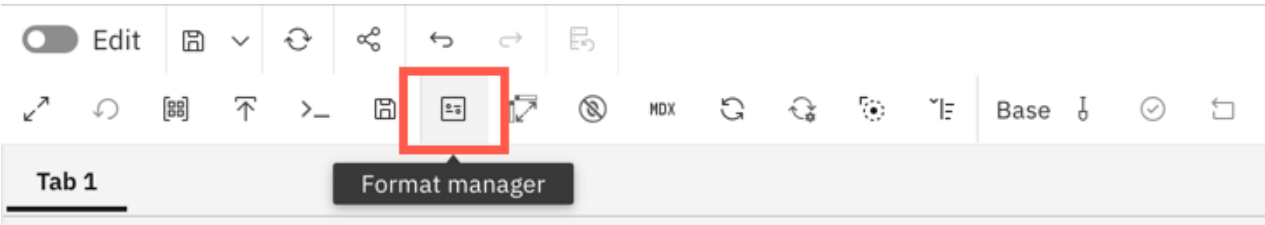
**Member**

- Data formats for data cells
- Font and string formats for data cells and header cells
- Member cell alignment
- Conditional rules for members

**Note:** You can inherit view formats and apply them to the member data or header cells.

## Accessing the Format manager

You can access the **Format manager** from the cube view toolbar. The **Format values** icon in the toolbar was repurposed for the **Format manager** in this release.



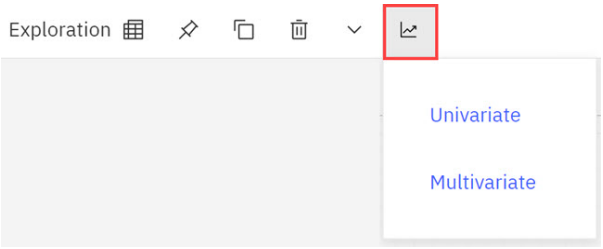

Right-click a member in the view and select **Member format** to jump right into the member formatting options.

**Note:** You can no longer clear applied formatting from the right-click menu. Instead, use **Format manager** or **Member format** to clear or change formats.

You can manage the availability of **Format manager** through the **Toolbar** icon settings in the **Properties** panel. Disable **Format manager** under **Properties > Custom > Toolbar** if you don't want users to see the **Format manager** icon in the cube viewer toolbar.

## Create multivariate forecasts

You can now create both univariate and multivariate forecasts in Planning Analytics Workspace.




A univariate forecast predicts the future values of a single variable based on historical data, using the Holt-Winters model.

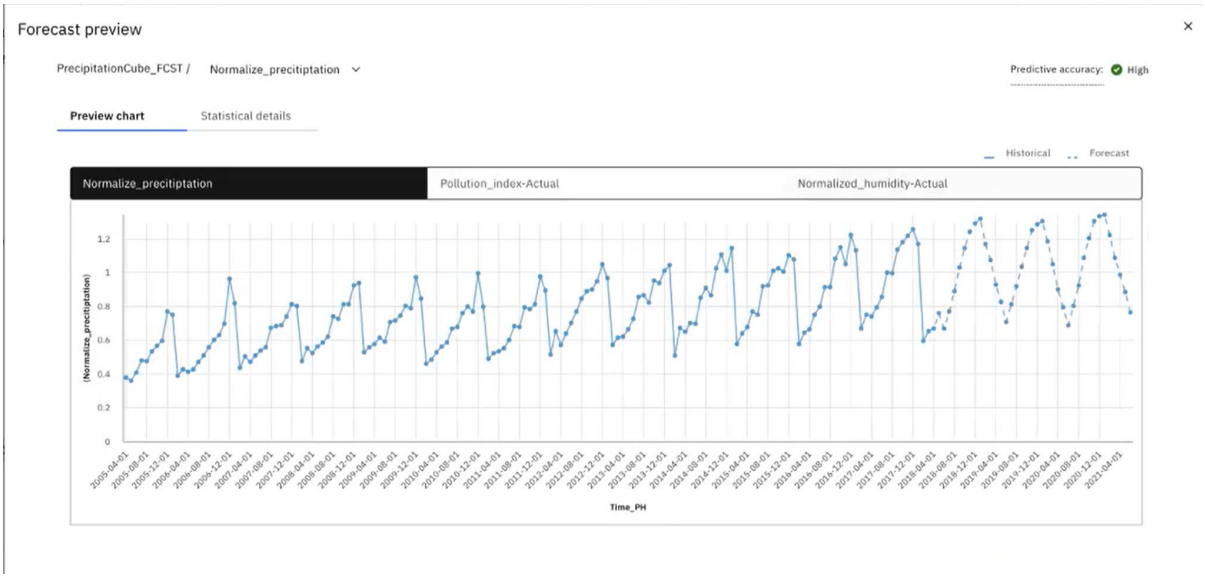
A multivariate forecast takes into consideration multiple time-dependent variables. The forecast model looks not only at historical data, but also the dependencies between the specified variables to create a forecast. The selected variables play a significant role in driving the forecast results. Therefore, it is important to select variables that have high correlation with the historical data.

Multivariate forecasting uses one of two models, vector auto-regressive (VAR) and auto-regressive integrated moving average (ARIMAX). If the historical data consists of less than 64 data points, VAR is used. If future values are included in the forecast, ARIMAX is used. If the historical data consists of more than 63 data points, but future values are *not* included in the forecast, then the optimal model is used. You have the option to include or exclude future values when you configure your forecast.

Note that neither multivariate model is seasonal. If your data is highly seasonal, then univariate forecasting is recommended.

The variables for a multivariate forecast can be selected from different cubes on the same Planning Analytics database, as long as both the historical cube and the variable cube use the same time dimensions. Time dimensions are the dimensions placed in the columns position of a view to represent a continuous set of data points.

When you generate and preview a multivariate forecast, visualizations are created for the member or members that are the target of the forecast, as well as the variables that contribute to the forecast calculation. For example, if you create a forecast for precipitation that considers pollution and humidity as influential variables, you'll get one visualization with historical and forecast values for precipitation. You'll also see separate visualizations for pollution and humidity that can help you understand how these variables influence the precipitation forecast.



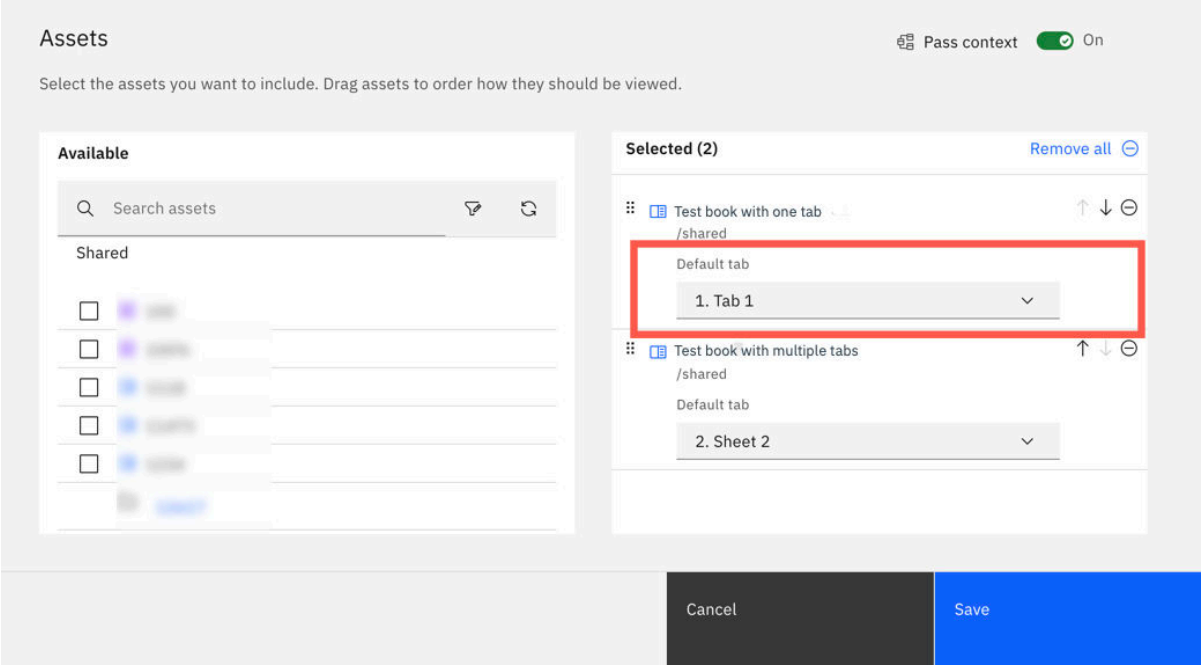

# What's new in plans and applications - 2.0.94

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.94.

## Open a specific tab as default in book assets

You can now set a tab to open by default in book assets in Applications and Plans.

To specify a default tab, make a selection under the new **Default tab** option on the **Assets** page when you attach a book to a section or a task.




When you click the book in a section or an application, the asset opens to the default tab that you selected.

# What's new in modeling - 2.0.94

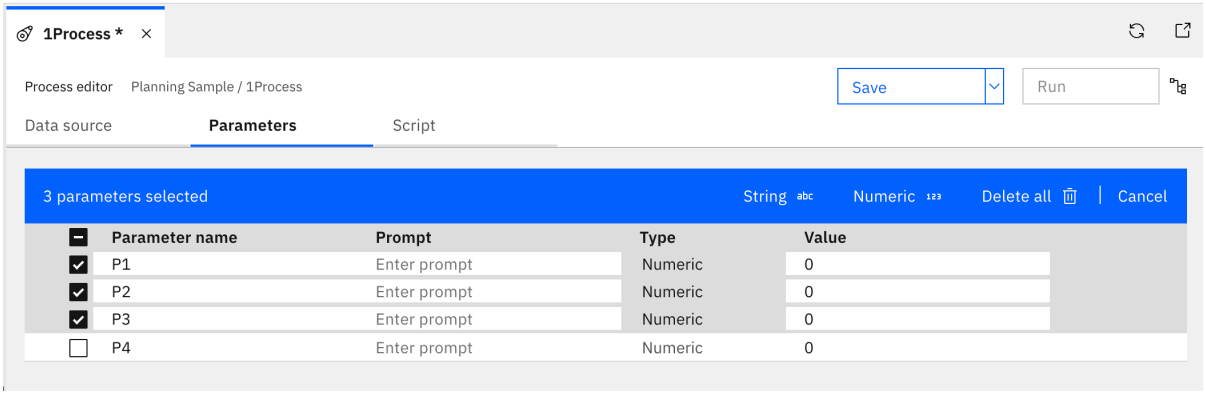
The following new and changed modeling features are available in Planning Analytics Workspace 2.0.94.

## Change multiple parameter types in the process editor

You can now change the type of multiple parameters in the process editor simultaneously.

To change the type of multiple parameters:

1. In **Data and Models** open a workbench.
2. In **Databases**, select a database and then click on a process.
3. In the **Process editor**, click the **Parameters** tab.
4. Click the check box for the parameters you want to change the **Type** of. You can also click the check box for **Parameter name** to select all the parameters in the list.
5. Click **String** abc or **Numeric** 123 to change all the selected parameters to that **Type**. Your changes will be reflected in the **Type** column.



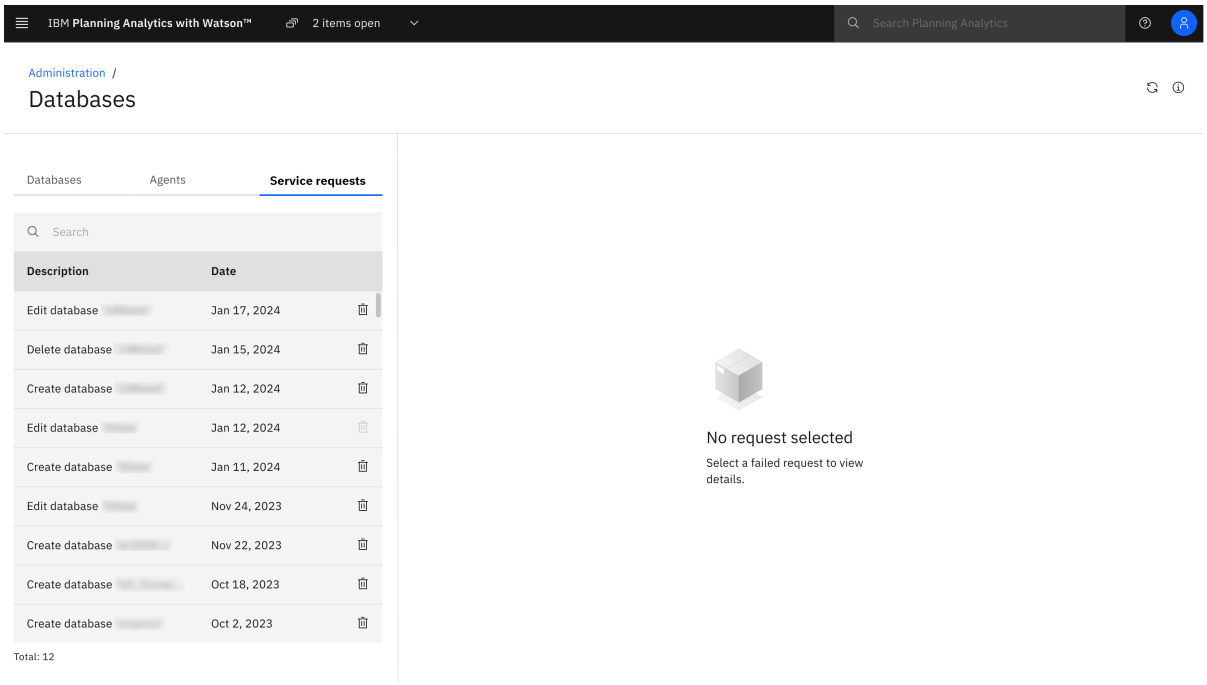
# What's new in administration - 2.0.94

The following new and changed administration features are available in Planning Analytics Workspace 2.0.94.

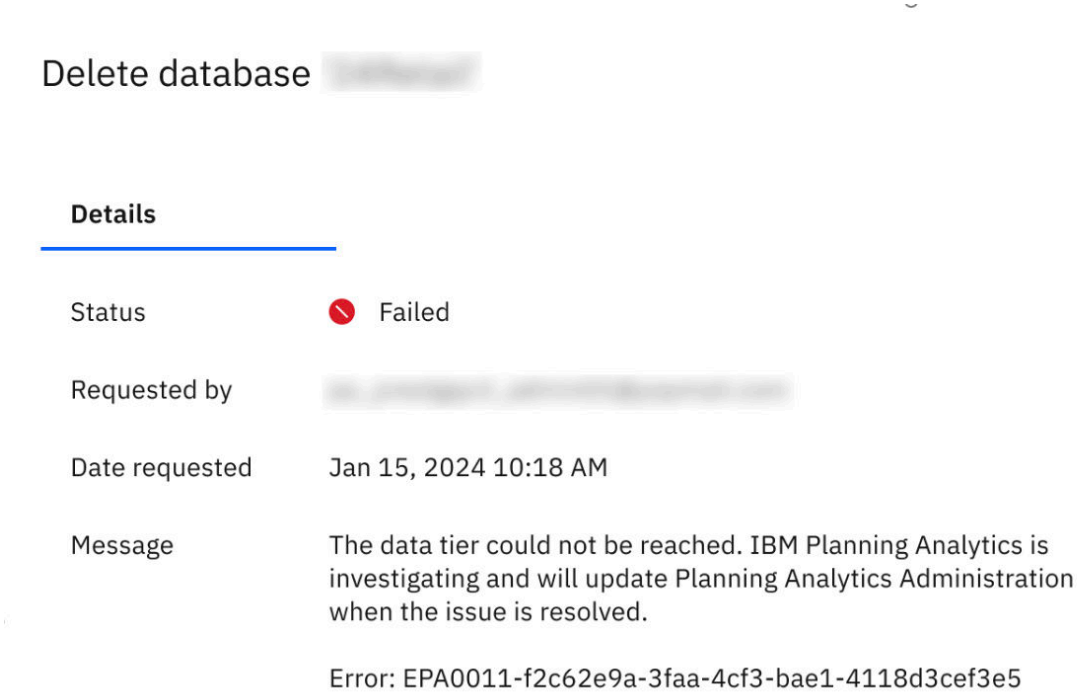
## View failed database requests in a separate list (Cloud only)

You can now view failed self-service database requests in IBM Planning Analytics Administration (Cloud only).

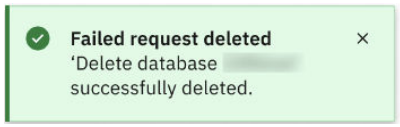
A new tab named **Service requests** is available on the **Databases** page. This tab contains a list of failed self-service database requests. You can sort the **Service requests** list by description or by the date the requests were created.




When you click a failed request, the **Details** panel displays the status of the request, who made the request, when the request was made, and why the request failed.




You can click the delete icon to delete a request on the **Service requests** tab. A notification appears when the request has been successfully deleted.

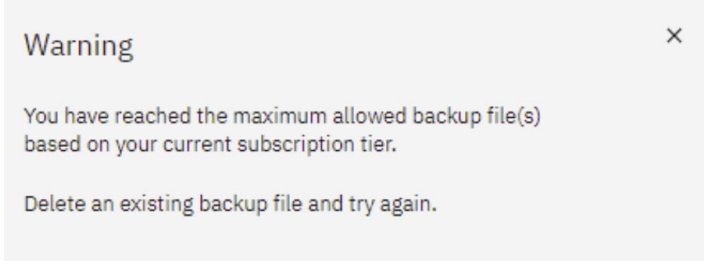



## Manual database backups limit enforced in Planning Analytics Administration (Planning Analytics as a Service only)

The number of manual database backups that you can save for a database is limited by your license tier.

| License tier | Maximum saved manual database backups |
|---|---|
| Essential | 2 |
| Standard | 5 |
| Premium | 10 |

If you attempt to save a manual backup when you already have the maximum backups saved, you are notified and requested to delete an existing backup.



## Deprecation notices - 2.0.94

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.94.

### End of support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 (Local only)

Support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 will end in June 2024.

This coincides with Red Hat's announced end of maintenance support for Red Hat Enterprise Linux 7. For more information, see https://www.redhat.com/en/blog/end-maintenance-red-hat-enterprise-linux-7-almost-here.

Planning Analytics Workspace Local versions 2.0.70 and later are supported on Red Hat Enterprise Linux version 8.

Planning Analytics Workspace Local versions 2.0.93 and later are supported on Red Hat Enterprise Linux version 9.

### End of support for Planning Analytics Workspace Local on Windows Server 2016 (Local only)

Support for Planning Analytics Workspace Local on Windows Server 2016 will end in 2024.

This is in accordance with Microsoft's announced end of Mainstream Support for Windows Server 2016 as of January 11, 2022. For more information, see https://learn.microsoft.com/en-us/lifecycle/products/windows-server-2016.

Planning Analytics Workspace Local versions 2.0.74 and later are supported on Windows Server 2019.

# 2.0.93 - What's new, February 20, 2024

Learn about new features and known issues in version 2.0.93 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

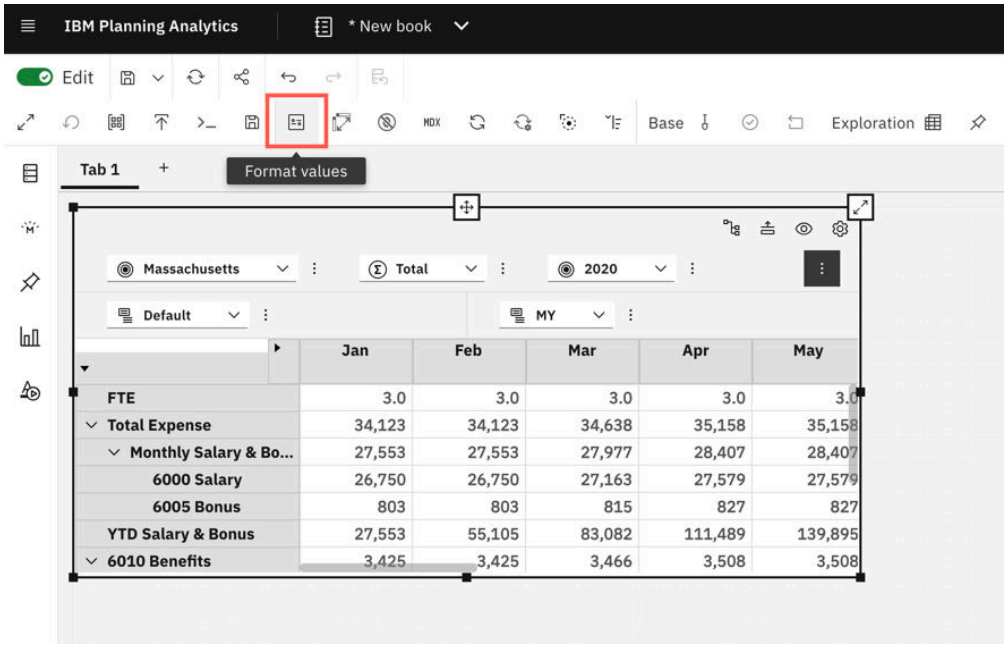
## What's new in books and reporting - 2.0.93

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.93.

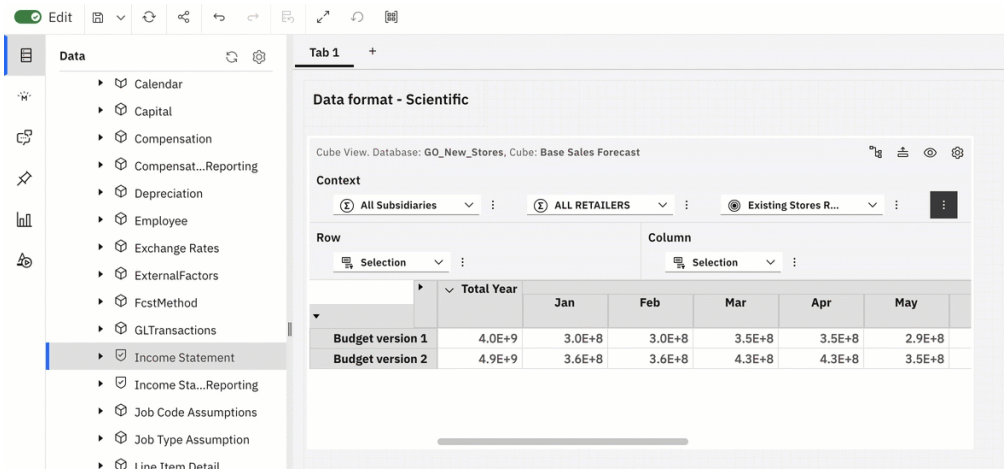
### Book retains data format when saved as a template

When you save a book as a template, any data format that you applied to an entire view or visualization now also saves to the template. When you create a new book using the template, the retained data format automatically applies to the view or visualization when you populate the data.

The data format is retained in a template only if you applied the format to an entire view or visualization by clicking **Format values** in the application toolbar.



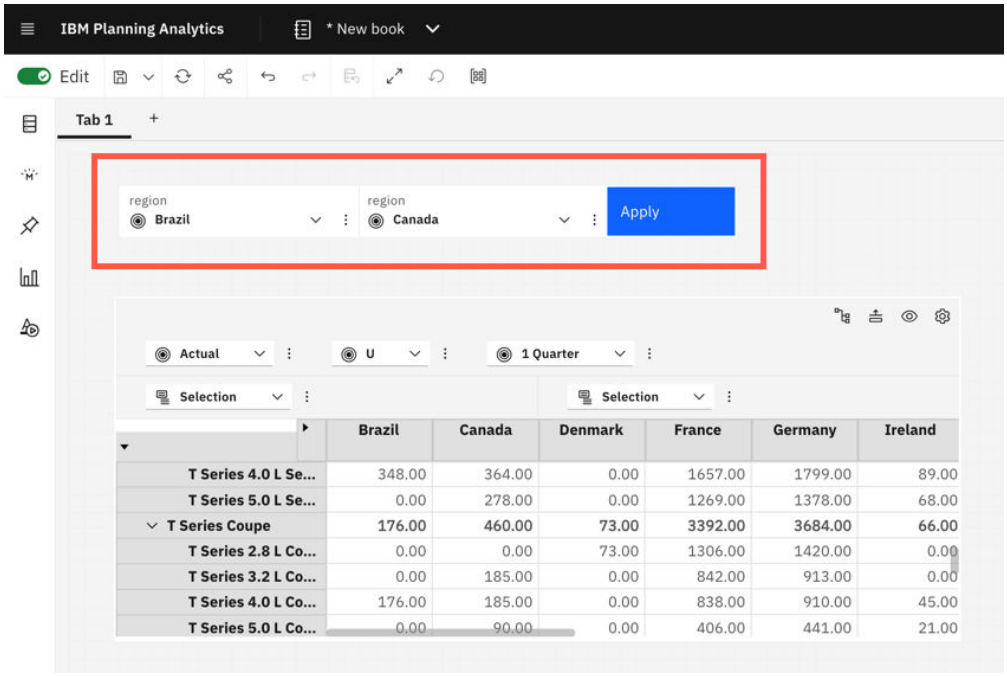

The data format is retained even when you replace the current view in the new book with another view.




For more information, refer to Save a book as a template and set it as default.

## Use the new advanced selector widget to synchronize members

The new Advanced selector widget supports one-way synchronization and has multiple selectors with the same hierarchy that lets you pick members from different groups to form a set. This new feature lets you easily compare members, view data for a range of members, or create a multiple selection list.




When you enable **Synchronize hierarchies** for a view or visualization, selecting a specific member in an Advanced selector updates the view or visualization. Changes in the Advanced selector widget trigger synchronization in the same synchronization group but does not synchronize to the changes of sets.

At any time, you can click **Undo** to reverse a selection change.

## Adding an Advanced selector widget

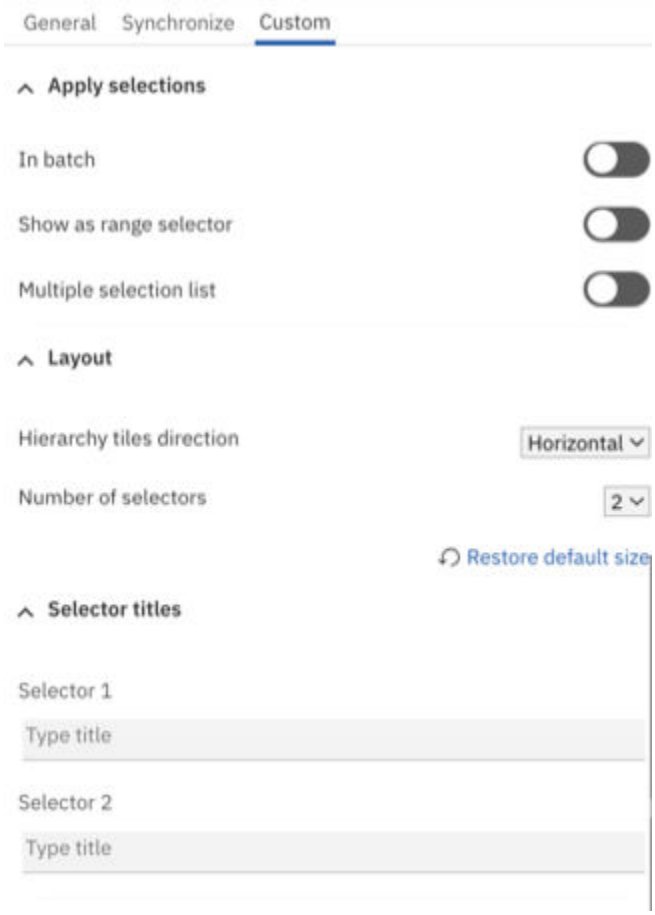
You can add Advanced selector widgets from an exploration row, column, or context area but not from benched dimensions.

1. In Edit mode, click the menu icon next to the dimension you want to create an advanced selector widget from.
2. Under **Add widget as**, select **Advanced selector**.

After you add an Advanced selector widget to your book, you can customize the widget in its appearance, layout, selection methods, and choose when selection changes apply to the synchronization group.

## Custom properties of an advanced selector widget

You can tailor the Advanced selector widget for your purposes.



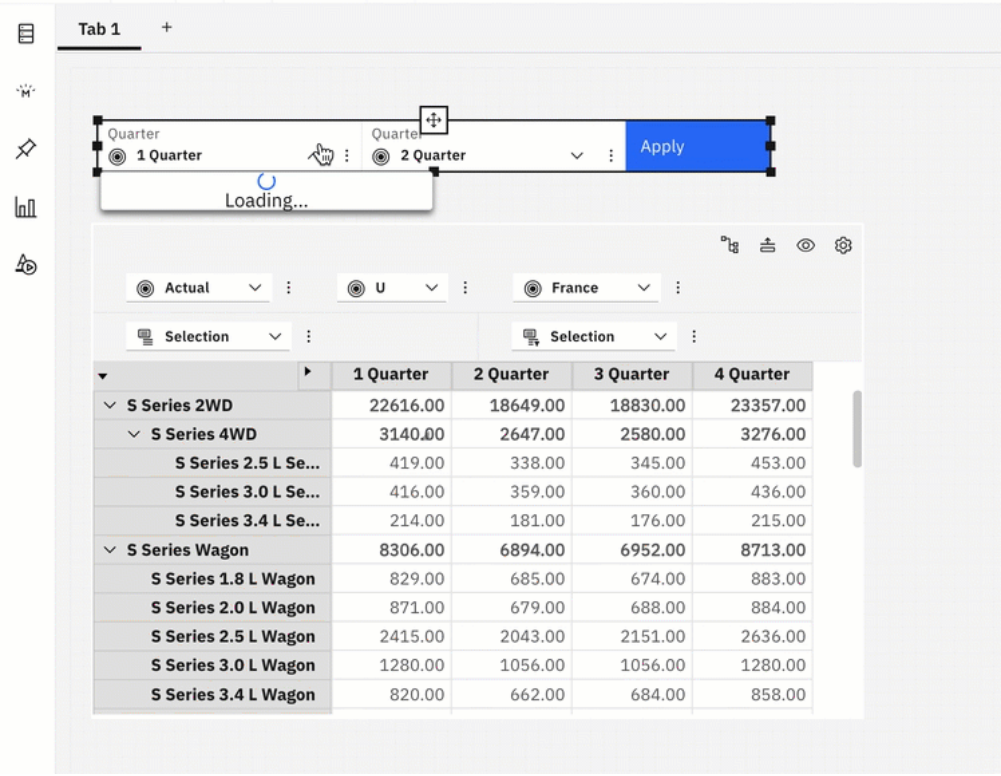

**Apply selections**

You can toggle if the change in one selection applies to its synchronization group immediately or changes from multiple selectors apply in a batch. Enable the **In batch** property if you want changes from multiple selectors to apply in a batch and only when the **Apply** button in the widget is pressed. Otherwise, any change in a member selection applies as soon as the change occurs.

Select **Show as range selector** if you want to see a range of members. The two selected members in the Advanced selector act as the first and last members of the member list and displays all the members between the two selections.

**Important:** The **Show as range selector** property works only when you have a maximum of two selectors in an Advanced selector widget.

In the following example, both the **In batch** and **Show as range selector** properties are enabled. When you change the member selections to **2 Quarter** and **4 Quarter** in the first and second selectors, the view updates to display all the members between **2 Quarter** and **4 Quarter**.




Enable **Multiple selection list** to display the Advanced selector widget as a list, which allows multiple member selections.

**Note:** You will need to also enable the **In batch** property when using **Multiple selection list** so that changes apply after all the selections are made.

**Layout**

In the **Layout** section, you can select whether you want the Advanced selector to display horizontally or vertically and you can restore the widget to its default size.

You can also select the number of selectors that display in an Advanced selector widget. You can choose between two to four selectors. By default, Advanced selector widgets contain two selectors.

**Selector titles**

You can specify the title for each selector in an Advanced selector widget.

**Selector style**

Much like the custom properties of the Selector widget, under the **Selector style** section, you can display an Advanced selector widget as a simple list, expand and collapse the widget, show the search bar, and much more.

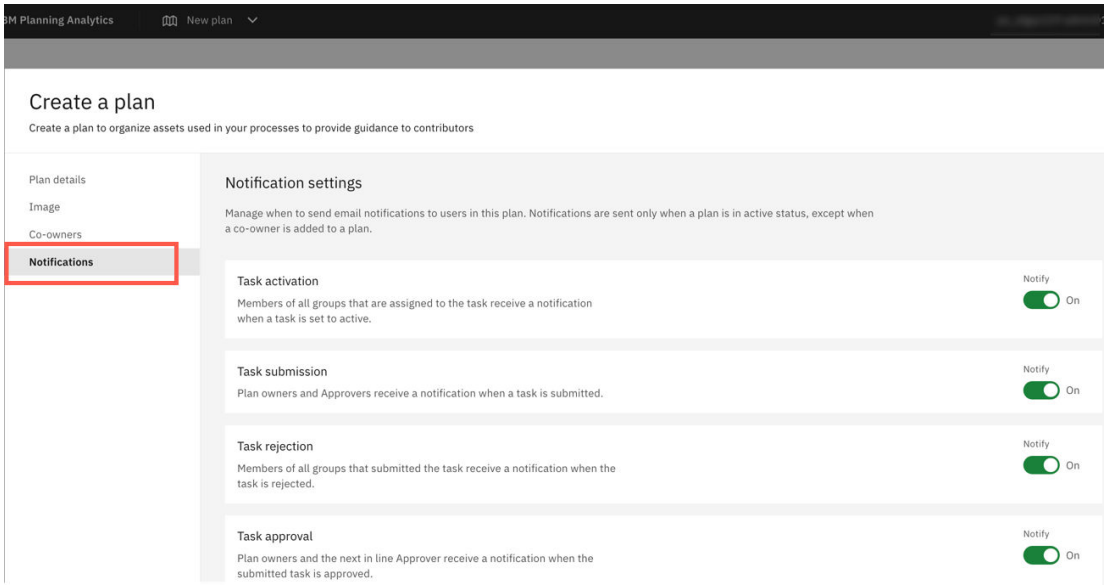
# What's new in plans and applications - 2.0.93

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.93.

## Manage email notifications in a plan

You can now turn on or turn off email notifications for individual events on the new **Notifications** tab in a plan. Previously, plan users automatically received email notifications when actions, such as a task submission or task approval, occurred in a plan.

When you create a new plan or edit an exiting one, you can manage notifications for individual events in a plan on the **Notifications** tab. By default, all notifications are turned on.

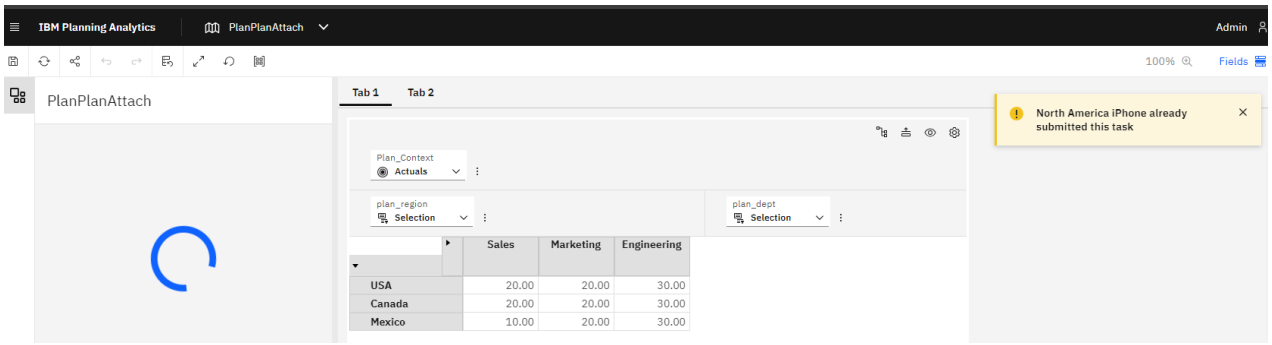



For more information, see Email notification in plans.

## Improved handling of simultaneous actions in Plans

Previously, when two users from the same group opened a plan to perform the same action at the same time, such as submitting or approving a task, both actions were recorded in the **Submission log**. Now, in IBM Planning Analytics Workspace 2.0.93, only the first action is logged in the **Submission log**.

When another user performs the same action, they receive a response that the action is already complete.




# What's new in modeling - 2.0.93

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.93.

## Copy and move multiple assets or folders

You can now copy and move multiple assets or folders to a new location on both the **Reports and Analysis** page and the **Data and Models** page in IBM Planning Analytics Workspace.

To copy or move multiple assets:

1. Click either the **Reports and Analysis** or the **Data and Models** tile on the Planning Analytics Workspace home page.
2. Click the check mark icon ☐ beside each of the assets or folders you want to copy or move to a different location.
3. Click **Copy or move** option in the toolbar.

4. Select a destination.
5. Click **Copy** or **Move**. The assets will be moved to the new location.

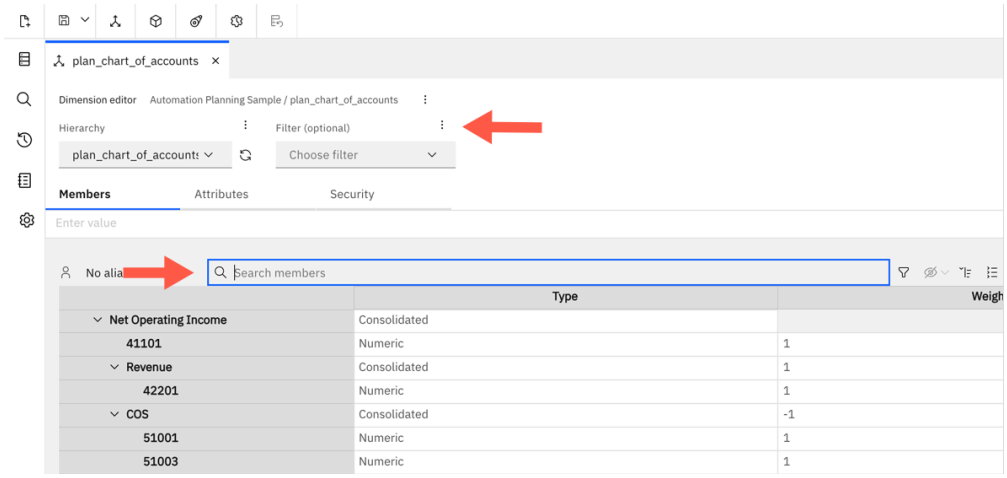
**Note:** If you copy or move an asset to the **Shared** folder, you are prompted to set user group permissions to the asset or folder.

If you copy or move an asset to a folder it is already located in, you are prompted to choose what to do with the duplicate asset.

## New filtering options available in the dimension editor

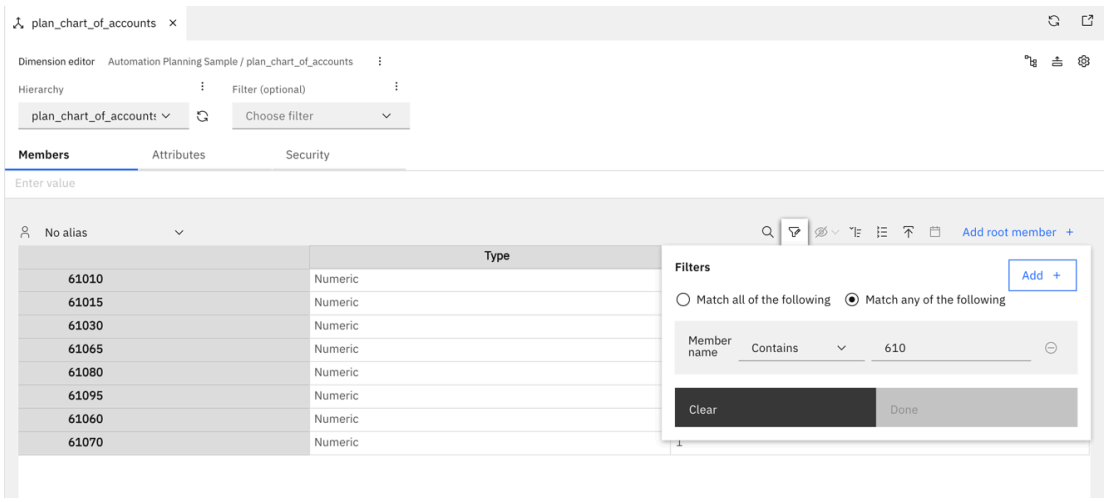
Additional filtering options are now available in the dimension editor, which allow you to further narrow the focus of members that are displayed in the editor.

Previously, you could filter members in the dimension editor with **Filter (optional)** or use **Search** to generate a list of members that match the search term.




In IBM Planning Analytics Workspace 2.0.93, along with the existing **Filter (optional)** and **Search** methods, new filtering options similar to those in the set editor are available under the new **Filter** icon.

Click the **Filter** icon in the dimension editor toolbar to filter on member names. You can choose operators such as = or < > to create dynamic selections and combine up to three filter parameters in one go.




When you apply these filters on the **Members** tab, the same filters automatically apply to the members on the **Attributes** tab.

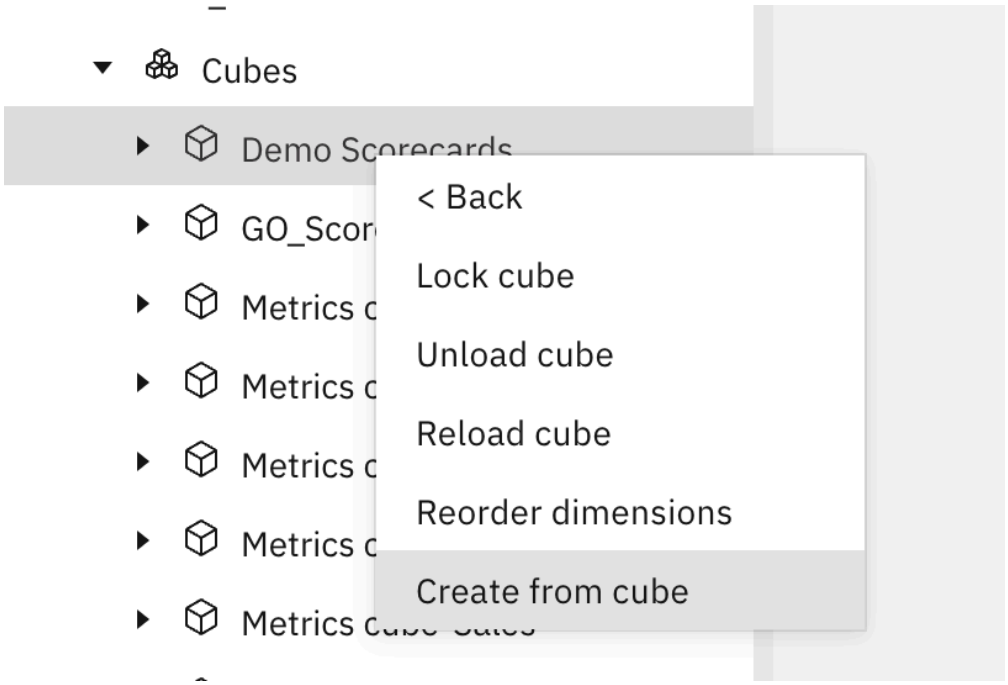
Clearing filters on **Members** or **Attributes** tab, clears the filters on both tabs.

## Replace an existing cube

You can now replace an existing cube with a new cube in the IBM Planning Analytics Workspace **Create cube** menu.

To replace an existing cube with a new cube, complete the following instructions:

1. In the **Databases** tree, right-click on the cube that you want to replace.
2. Click **Create from cube**.




3. In the **Create cube** menu, you can change the database and name of the cube. Then, click the check box next to **Replace cube**.
4. Click **Next**.
5. In the **Dimensions** menu, you can add or remove dimensions from the cube you are creating.
6. Click **Create**. A warning will appear, explaining that deleted data is not recoverable.
7. Click **OK**. The existing cube is deleted, and a new one is created with the same name.

For more information on creating cubes, see: Create a cube.

## Map to existing dimensions when importing data to a new cube

You can now map to an existing dimension or control dimension when you import data to a new cube.

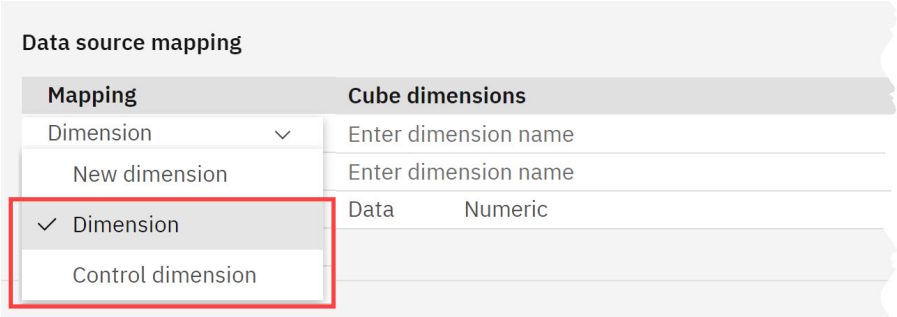
Previously, importing data to a new cube required that all cube dimensions be created as new from the imported data.

When importing data in Planning Analytics Workspace 2.0.93 and later, the **Data source mapping** section includes three options for mapping dimensions:

**New dimension**
: Data in the source is mapped to a new dimension. You must provide a unique name for the new dimension.

**Dimension**
: Data in the source is mapped to an existing dimension. You must select from a list of existing dimensions on the **Cube dimensions** column.

**Control dimension**
Data in the source is mapped to an existing control dimension. You must select from a list of existing control dimensions on the **Cube dimensions** column.



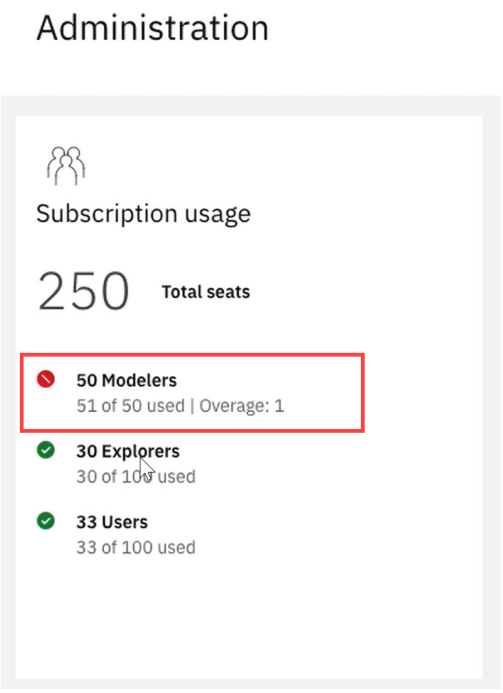

# What's new in administration - 2.0.93

The following new and changed administration features are available in Planning Analytics Workspace 2.0.93.

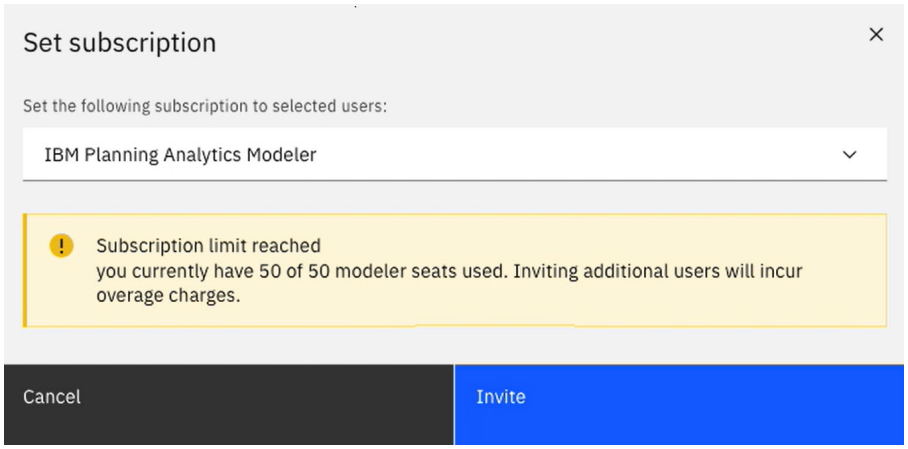
## View subscription overages in Planning Analytics Administration on Cloud

Planning Analytics Workspace subscription administrators can now view subscription overages directly in Planning Analytics Administration without accessing IBM Subscriptions Management.

The **Subscription usage** tile on the Administration page shows the total number of seats available for your subscription plan, as well as the number of seats used and available for each category of user. If an overage exists for any category it is reported on the tile.




Additionally, you are now warned if an attempt to set a subscription for one or more users will result in an overage. In such an event, you can click **Cancel** to avoid overage charges.




## Planning Analytics Workspace Local support for Red Hat Enterprise Linux 9 (Local only)

You can now install Planning Analytics Workspace Local version 2.0.93 or later on Red Hat Enterprise Linux 9.

For details, see Installing Planning Analytics Workspace Local.

## Improved availability of manual database backups in Planning Analytics Workspace on Planning Analytics as a Service

A new location for manual database backups in Planning Analytics Workspace on Planning Analytics as a Service makes it easier to access and use backups.

Prior to Planning Analytics Workspace 2.0.93, manual database backups were stored in the `.backupsets` directory of File Manager on the Modeling Workbench, which is within the Planning Analytics Workspace database. This location made it impossible to access manual backups when the database was unavailable.

Conversely, automated database backups were stored to a location external to the database, which remains accessible even when the associated Planning Analytics Workspace database is unavailable.

As of Planning Analytics Workspace 2.0.93, **both** manual and automated database backups in Planning Analytics Workspace on Planning Analytics as a Service are stored in a location external to the database. All manual backups created in Planning Analytics Workspace 2.0.93 or later and all automated backups appear on the database **Backups** tab, regardless of the current state of the database.

This change means that manual backups created prior to Planning Analytics Workspace 2.0.93 and stored in File Manager no longer appear on the **Backups** tab. You can, however, continue to access any such backups through File Manager.

If you'd like to move a manual backup from File Manager to the storage location external to the database:

1. Locate and select the manual backup (`tar.gz` archive) in File Manager on the Modeling Workbench.
2. Click the **Action** button ⋮ for the backup, then click **Download**. Save the backup archive to a convenient location.
3. Return to the **Databases** page in Planning Analytics Administration.
4. Click the database that is associated with the backup.
5. Click **Upload** ⤒.
6. Specify the backup archive that you saved in Step 2.
7. Click **Upload**.

The uploaded archive appears on the **Backups** tab as a manual backup and the **Date created** reflects the date on which the archive was uploaded to the Simple Storage Service location.

## Deprecation notices - 2.0.93

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.93.

### Selector lists and Selector tile widgets are converted to Selector visualizations in Planning Analytics Workspace 2.0.93

Any Selector list or Selector tile widget present in books is converted to a Selector visualization in Planning Analytics Workspace 2.0.93.

The Selector visualization is styled to most closely replicate the list or tile widget it replaces.

Selector visualizations were originally introduced in Planning Analytics Workspace 2.0.89, but were subject to administrator enablement. As of Planning Analytics Workspace 2.0.93, Selector visualizations are enabled by default as the only type of Selector available in all environments. You can no longer create Selector lists or Selector tile widgets.

The Selector visualization behaves much as Selector lists and Selector tile widgets did, but there are multiple custom visualization properties that you can use to better tailor a Selector visualization for your purposes. There are also several ways to insert a Selector visualization into a book.

For more information, see Create a Selector visualization to manage views.

### End of support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 (Local only)

Support for Planning Analytics Workspace Local on Red Hat Enterprise Linux 7 will end in June 2024.

This coincides with Red Hat's announced end of maintenance support for Red Hat Enterprise Linux 7. For more information, see https://www.redhat.com/en/blog/end-maintenance-red-hat-enterprise-linux-7-almost-here.

Planning Analytics Workspace Local versions 2.0.70 and later are supported on Red Hat Enterprise Linux version 8.

Planning Analytics Workspace Local versions 2.0.93 and later are supported on Red Hat Enterprise Linux version 9.

### End of support for Planning Analytics Workspace Local on Windows Server 2016 (Local only)

Support for Planning Analytics Workspace Local on Windows Server 2016 will end in 2024.

This is in accordance with Microsoft's announced end of Mainstream Support for Windows Server 2016 as of January 11, 2022. For more information, see https://learn.microsoft.com/en-us/lifecycle/products/windows-server-2016.

Planning Analytics Workspace Local versions 2.0.74 and later are supported on Windows Server 2019.

# 2.0.92 - What's new, December 12, 2023

Learn about new features and known issues in version 2.0.92 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

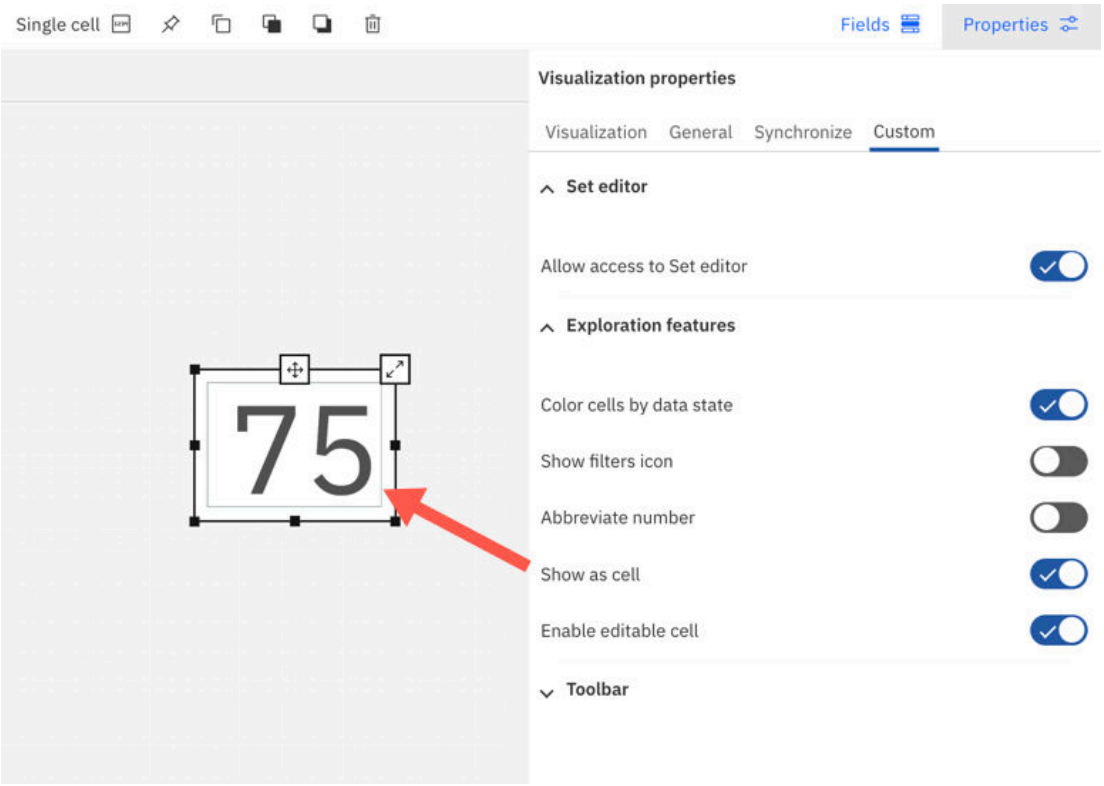
# What's new in books and reporting - 2.0.92

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.92.

## Changes to single cell widget properties

The **Show as cell** property for single cell widgets has been updated to control only the display of an outline around values in single cell widgets. A new property, **Enable editable cell**, now allows modification of values in single cell widgets. Previously, you could modify values in single cell widgets with the **Show as cell** property but couldn't remove the outline around values in the widgets.

Enable the **Show as cell** property under **Properties > Custom > Exploration features** to display an outline around the value in a single cell widget.




To modify values in a single cell widget, turn on the **Enable editable cell** property under **Properties > Custom > Exploration features** .

Both, the **Show as cell** and **Enable editable cell** properties, are turned on by default in new single cell widgets.

When you upgrade to Planning Analytics Workspace 2.0.92, if an existing single cell widget is set to **Show as cell** , then both the **Show as cell** and **Enable editable cell** properties are enabled in 2.0.92.

# What's new in plans and applications - 2.0.92

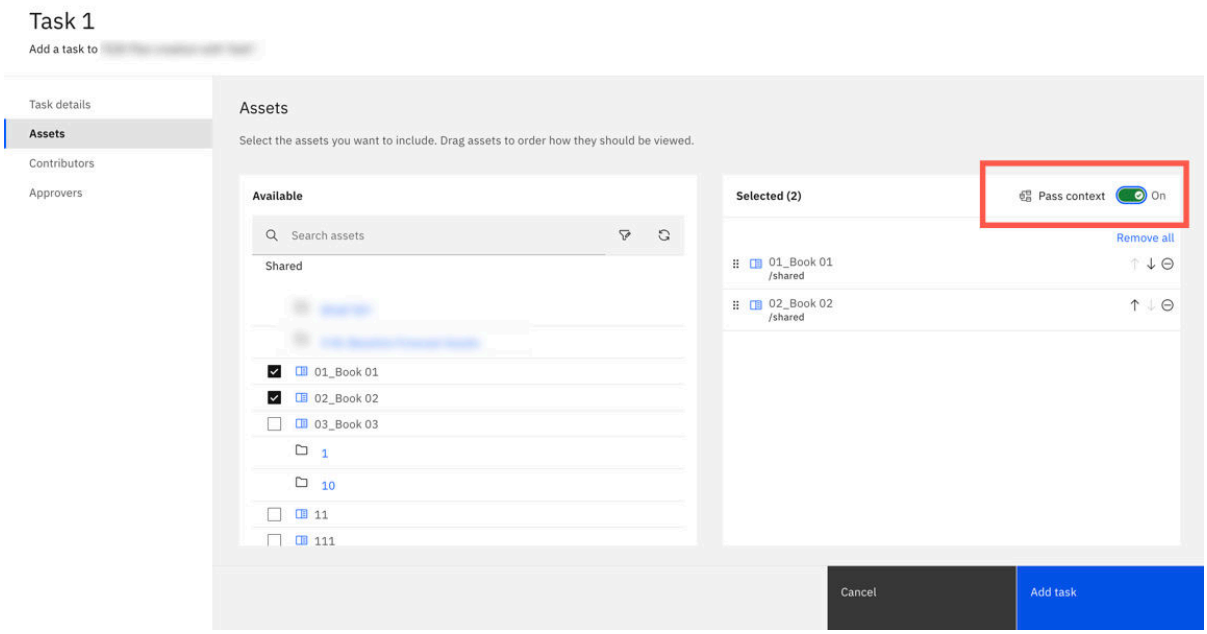
The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.92.

## Pass context between books in an application or plan

You can now pass context between books that are added as assets in an application or a plan that use the same synchronization group.

When you add books to sections in an application or tasks in a plan, enable the new **Pass context** option to pass context between any books that have synchronization enabled and use the same synchronization group.

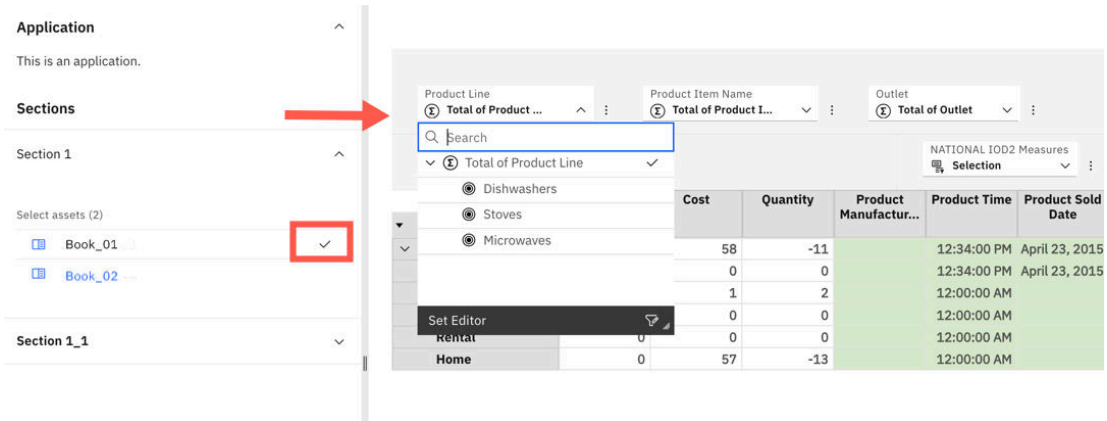
**Remember:** Before you can pass context between books of the same application or plan, you need to first setup synchronization in these assets in **Reports and Analysis**.



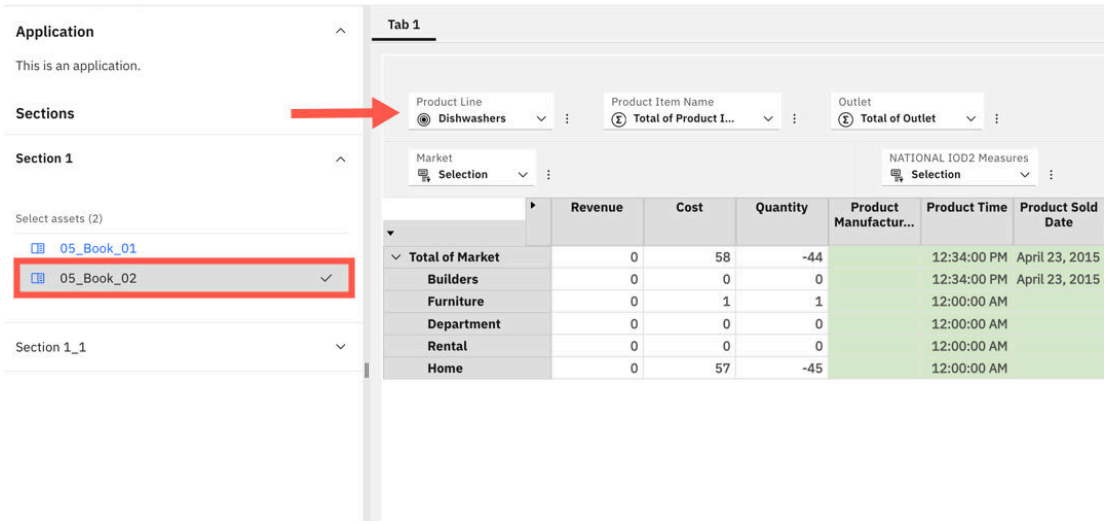

The **Pass context** feature supports all three synchronization groups (Book, Sheet, and Custom) as well as synchronization of hierarchies and gestures.

When you pass context between books that are synchronized at the book level, the source book is the one that opens first and has a check-mark next to its name in the application section or plan task. The target book is the book that you click on next.

In the following example, Book_01 and Book_02 are synchronized at the **Book** level. Book_01 is the source book in application Section 1.




When you change the **Product Line** to **Dishwashers** in Book_01 and go to Book_02, the hierarchy in Book_02 automatically updates to **Dishwashers**.




In books that are synchronized at the sheet level, you can choose the source tab but you cannot choose the target tab. The target tab is always the first tab of whichever book you click on after you make changes to the source book.

Similarly, in custom group level synchronization, widgets that belong to the same group will be synchronized together when you enable **Pass context** for books in an application or plan. As members of a hierarchy that display in one widget in the source book change, the same members display in synchronized widgets that use the same hierarchy in the target book.

# What's new in modeling - 2.0.92

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.92.

## Process editor improvements

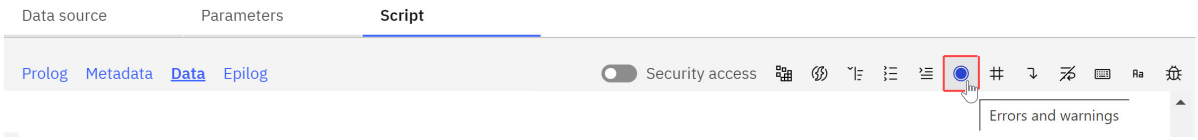
The **Script** tab on the process editor in Planning Analytics Workspace 2.0.92 introduces several performance and usability improvements that make it easier to create and manage your process scripts.

### Improved scrolling

Scrolling is now more responsive for scripts that exceed a single page view, resulting in a performance improvement over previous versions.

### Error and warning reporting

The toolbar on the process editor includes a new **Errors and warnings** icon.




Click **Errors and warnings** to reveal an error and warning panel at the bottom of the process editor. Click any warning or error in the panel to go directly to the line in your process script where the error or warning occurs.

### Select all lines within a script section

A process script contains four distinct sections: Prolog, Metadata, Data, and Epilog.

When you press CTRL+A (select all), all lines within the section where the cursor is positioned are selected.

This is a change of behavior from previous versions of the process editor, where CTRL+A selected all lines across all sections of the process script.

### Filter variables during debugging

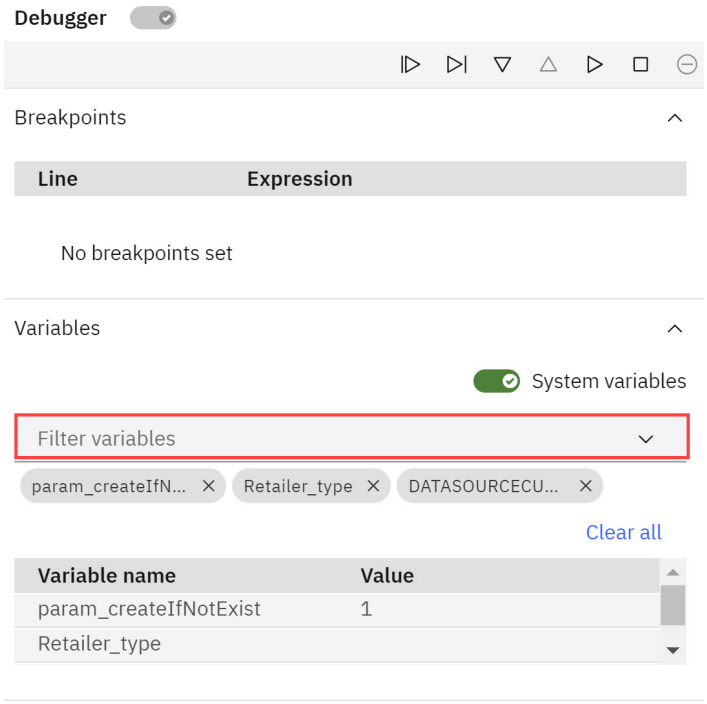
You can now use a filter to examine only selected variables while debugging a process.

The Variables section of the Debugger includes a **Filter variables** option. Click **Filter variables** and select a variable that you want to include in the debugging process. You can specify multiple variables for inclusion in debugging, but you must click **Filter variables** for each variable that you want to include.

If you do not set a filter to include at least one variable for inclusion during debugging, *all* variables are examined during debugging.

By default, only local process variables are shown when you click **Filter variables**. To include system variables for selection, toggle **System variables** to the **On** position.

This example shows the Debugger configured to include three variables in debugging.




## What's new in administration - 2.0.92

The following new and changed administration features are available in Planning Analytics Workspace 2.0.92.

### Add an individual user in Planning Analytics Administration Local (Local only)

A new UI option lets you add an individual user in Planning Analytics Administration Local.

#### About this task

You can continue to upload a `.csv` file to add multiple users to Planning Analytics Workspace Local.

#### Procedure

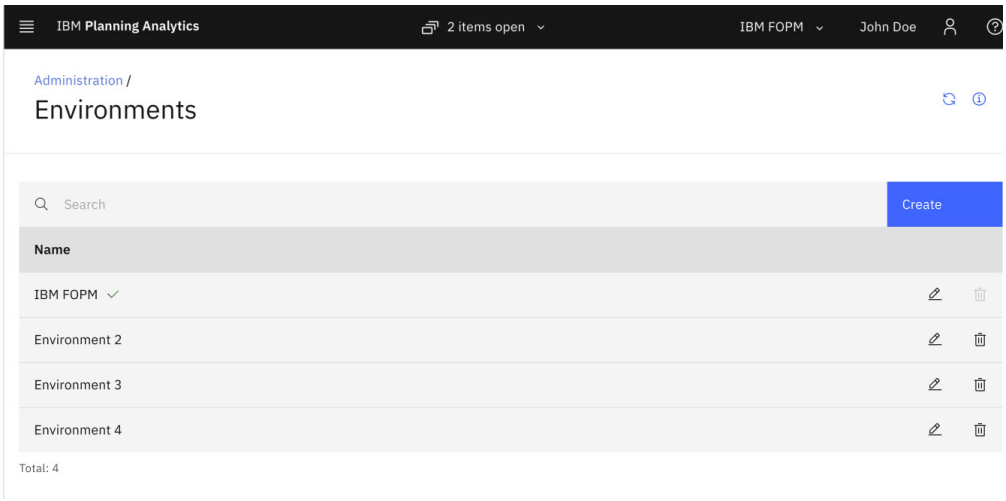
1. Click the **Administration** tile on the Planning Analytics Workspace home page.
2. Click the **Users and Groups** tile.
3. Click the **Users** tab.

4. Click **Add**.
5. Enter the required information for the new user, then click **Add**.

## Create and edit multiple Planning Analytics Local environments (Local only)

You can manage multiple environments in Planning Analytics Administration Local.

**Important:** You must have Environments admin privileges to create, edit, or delete environments.




To create a new environment:

1. Enable multiple environments by adding the following to the `paw.env` on Linux or `paw.ps1` file on Windows:

   ```
   export ENABLE_MULTI_ENV="true"
   ```

2. Restart Planning Analytics Workspace.
3. Click the **Administration** tile on the Planning Analytics Workspace home page.
4. Click the **Environments** tile.
5. Click **Create**.
6. Enter an **Environment name**, **Spreadsheet service URL**, and **Admin host**, then click **Create**.

   The **Admin host** that you specify determines which databases are available in the environment. Only the databases registered on the specified admin host are available in the environment.

   You can choose to leave the **Spreadsheet service URL** and **Admin host** values blank when you create an environment. Entries for these values are then automatically used from the `paw.env` file on Linux or `paw.ps1` file on Windows.

To edit or delete an existing environment, click the environment name on the **Environments** page, then click the appropriate icon.

You can assign an individual user to any combination of available environments. Users can have different roles in different environments. For example, a modeler in Environment 1 could be assigned the analyst role in Environment 2.

When you log in to Planning Analytics Local and have access to multiple environments, an environment switcher is available on the Planning Analytics Workspace Local home page, adjacent to your user name. You can click the switcher to switch between environments that are available to you.




**Tip:** If you have multiple environments available to your users, consider applying a custom theme to each environment to make it easier to visually differentiate between environments.

### Connecting to Planning Analytics Workspace URL in Planning Analytics for Excel

You can connect to a multi-tenant Planning Analytics Workspace URL in Planning Analytics for Microsoft Excel. A multi-tenant connection URL might look like this:

**`https://myexampleserver.com/?accountId=Z7IGUHW0LBYE&tenantId=GDY324GZA2FU`**

If you are using **Cognos Authentication Manager (CAM) authentication** (mode 5), when you connect to a Planning Analytics Workspace URL with a tenantID in Planning Analytics for Excel, the data source selection is empty. To connect to the URL in Planning Analytics for Excel, enter the URL without the accountID and tenantID, then select the tenant in the tenant selector.

### Important note on upgrading to Planning Analytics Workspace Local 2.0.92 - Administrator roles

Planning Analytics Workspace Local now supports multiple environments.

When you upgrade to 2.0.92, you need to assign Environments admin privileges to at least one user with an Administrator role. You can also promote a non-admin user to an Administrator role and assign them the Environments admin privileges using the command line.

For more information, refer to Promote a user to administrator (Local only).

## Deprecation notices - 2.0.92

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.92.

### Classic dimension editor removed from Planning Analytics Workspace 2.0.92

As of release 2.0.92, the classic dimension editor is no longer available in Planning Analytics Workspace.

All options for **Edit dimension (Classic)** and **Edit hierarchy (Classic)** have been removed from the software.

The **Edit dimension** and **Edit hierarchy** options remain available to open dimensions or hierarchies in the new experience dimension editor.

### Selector lists and Selector tile widgets are converted to Selector visualizations in Planning Analytics Workspace 2.0.93

Any Selector list or Selector tile widget present in books is converted to a Selector visualization in Planning Analytics Workspace 2.0.93.

The Selector visualization is styled to most closely replicate the list or tile widget it replaces.

Selector visualizations were originally introduced in Planning Analytics Workspace 2.0.89, but were subject to administrator enablement. As of Planning Analytics Workspace 2.0.93, Selector visualizations are enabled by default as the only type of Selector available in all environments. You can no longer create Selector lists or Selector tile widgets.

The Selector visualization behaves much as Selector lists and Selector tile widgets did, but there are multiple custom visualization properties that you can use to better tailor a Selector visualization for your purposes. There are also several ways to insert a Selector visualization into a book.

For more information, see Create a Selector visualization to manage views.

# 2.0.91 - What's new, November 14, 2023

Learn about new features and known issues in version 2.0.91 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.91

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.91.

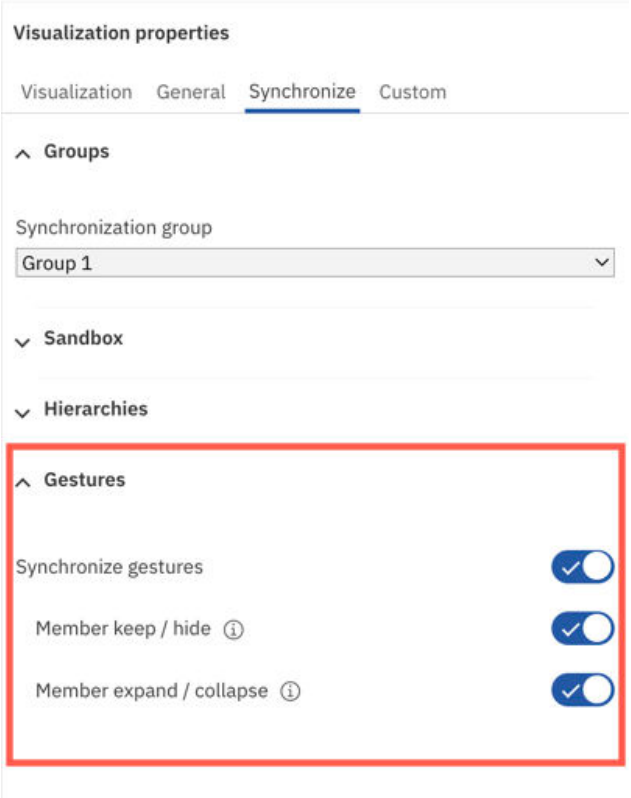
### Synchronize gestures across widgets

You can now synchronize certain gestures across widgets such as explorations and visualizations.

Much like synchronizing hierarchies, enabling synchronization for a gesture allows the widget to receive the synchronization event. You can use this feature to build a group with widgets that receive different gestures from other widgets in the group.

You can synchronize the following gestures under **Properties > Synchronize > Gestures**.

- Expand/Collapse
- Keep
- Hide (and Unhide all)




**Member keep / hide**

Use **Member keep / hide** to keep or hide the hierarchy in multiple widgets. A hierarchy might be on a row in one widget and a column in another. When you synchronize **Keep** or **Hide**, the widgets update based on where the synchronized hierarchy is located in the widget.

**Member expand / collapse**

Use **Member expand / collapse** to expand or collapse a consolidated hierarchy in multiple widgets. A hierarchy might be on a row in one widget and a column in another. When you synchronize **Expand/ Collapse**, the widgets update based on where the synchronized hierarchy is located in the widget.

You can synchronize only those gestures that are symmetric. For example, if you used **Target selection** (or **F7**) to make asymmetric selections on rows and columns, you won't be able to synchronize the widgets using **Member keep / hide**. Similarly, if you expanded or collapsed rows and columns asymmetrically, you can't use **Member expand / collapse** to synchronize the widgets.

**Note:** Synchronization gestures are not available for synchronized tiles and websheets. You can synchronize gestures in selector widgets if the widgets are showing a list of members.

For more information, see Synchronize objects in a book or sheet.

## Setting synchronization for gestures

1. Select the widgets that you want to synchronize and click **Properties**.
2. Select **Synchronize** and under **Hierarchies**, enable **Synchronize hierarchies** to enable synchronization.
3. Expand the **Gestures** section and enable **Synchronize gestures**.
   a. To keep or hide a row or column (including unhiding all), enable **Member keep / hide**.
   b. To expand or collapse a member, enable **Member expand / collapse**.

## Improvements to alignment of single cell widgets

When you create a new single cell widget from the cube viewer, the cell alignment of the widget now matches that of the source cell in the cube viewer. The direction in which text is added when you type in the widget (type-in direction), now also matches the cell alignment of the cube viewer.

For example, if the source cell in a cube viewer is left-aligned, any new single cell widget that you create from that source cell is left-aligned. The type-in direction of the single cell widget is also left-aligned.

**Important:** The cell alignment update applies to only new single cell widgets. Cell alignment in existing single cell widgets will not change to match the source cell. However, the type-in direction of existing cell widgets was updated to match the source cell's alignment.

Even if a source cell in the cube viewer uses the default alignment and doesn't have a cell alignment selected (under **Cells alignment**), the alignment, and type-in direction of a new single cell widget matches that of the source cell. The default cell alignment is left-aligned for string cells and right aligned for numbers.

After a single cell widget is added to a book, any changes you make to the cell alignment of the source cell, does not apply to the alignment or type-in direction of the single cell widget.

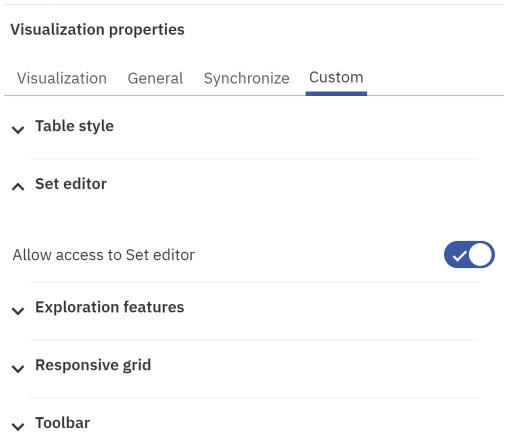
Previously, when you created single cell widgets from the cube viewer, the values in single cell widgets were centered and the type-in direction was right aligned.

## Set editor access in Properties

You can now enable and disable access to the set editor for visualizations and explorations in **Properties**.

Before, users edited the current set by accessing the set editor from the visualization or exploration. Now, you can modify access to the set editor that hides options to interact with the set editor.

You can apply this setting by using the **Allow access to Set editor** toggle. You can find this setting in the **Properties** panel, then in the **Custom** tab open the **Set editor** section.

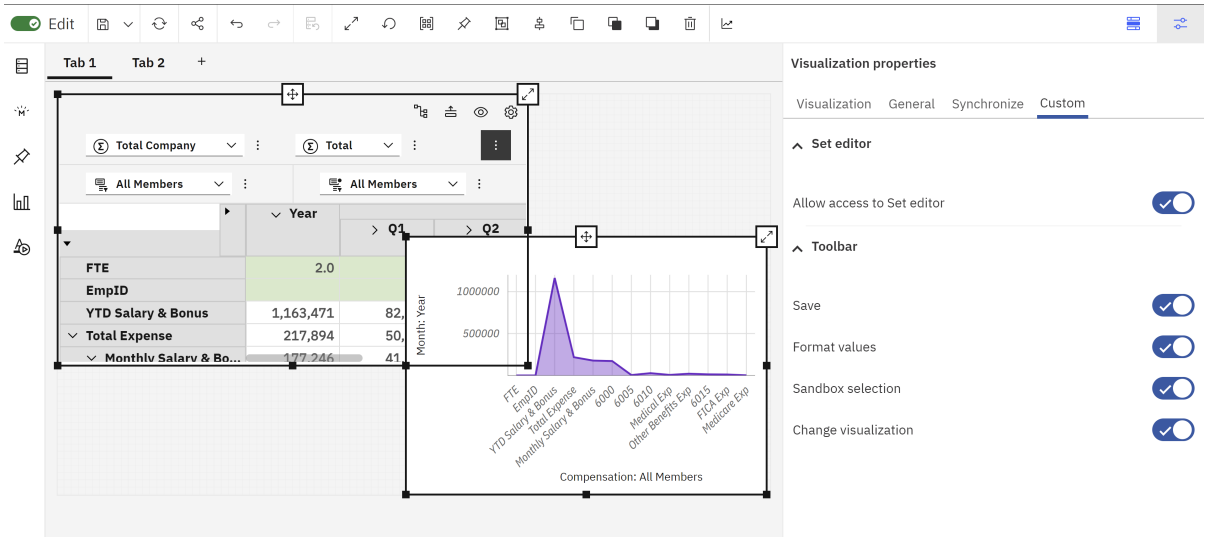



When you turn off the **Allow access to Set editor** toggle, all set editor menu options are hidden in all visualizations and explorations. In a cube viewer, the menu options **Open set editor**, **Enter named set**, and **Customize sets** are hidden if you click the More icon ⋮ for a dimension or member.

For a visualization, such as Area, you can right-click on the x-axis and y-axis labels to see that the menu options **Edit this set** and **Customize sets** are hidden. Alternatively, in the **Fields** panel, you can click the More icon ⋮ in the x-axis and y-axis fields to see that the same menu options are hidden. Also, under the **Filters** section you can notice that the **Edit this set** menu option is hidden for each filter if the toggle is off.

The setting of the **Allow access to Set editor** toggle stays the same even if you change the visualization to a different one. If you make multiple selections of visualizations, then the common properties, which are the **Set editor** and **Toolbar** properties, are available. When you change the set editor access, then the setting is applied to all the selected visualizations. The **Toolbar** properties apply to all selections of

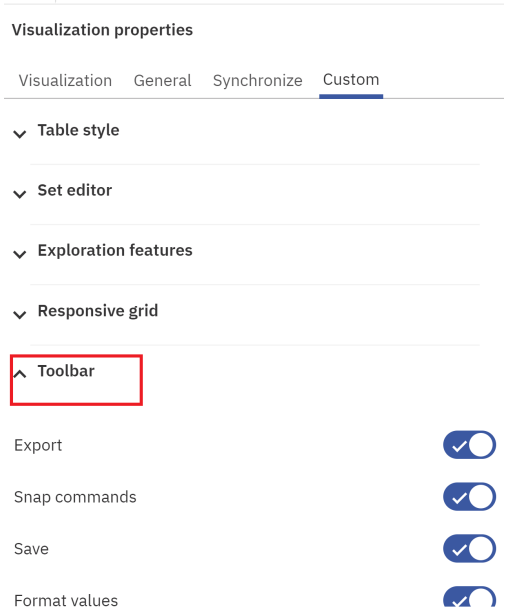
visualizations as well. For more information on **Toolbar** properties, see New toolbar properties for the cube viewer and visualizations.




## Toolbar properties for the cube viewer and visualizations

You can now enable and disable toolbar icons for the cube viewer and visualizations in the **Properties** panel. Managing which icons are available on the toolbar gives you the ability to control which features are available to your users and helps you to create books with a focused experience.

Toolbar icons are managed by applying the **Toolbar** icon settings in the **Properties** panel. You can find the settings on the **Custom** properties tab, under the **Toolbar** section.




For a cube viewer, you can enable or disable these nine icons:

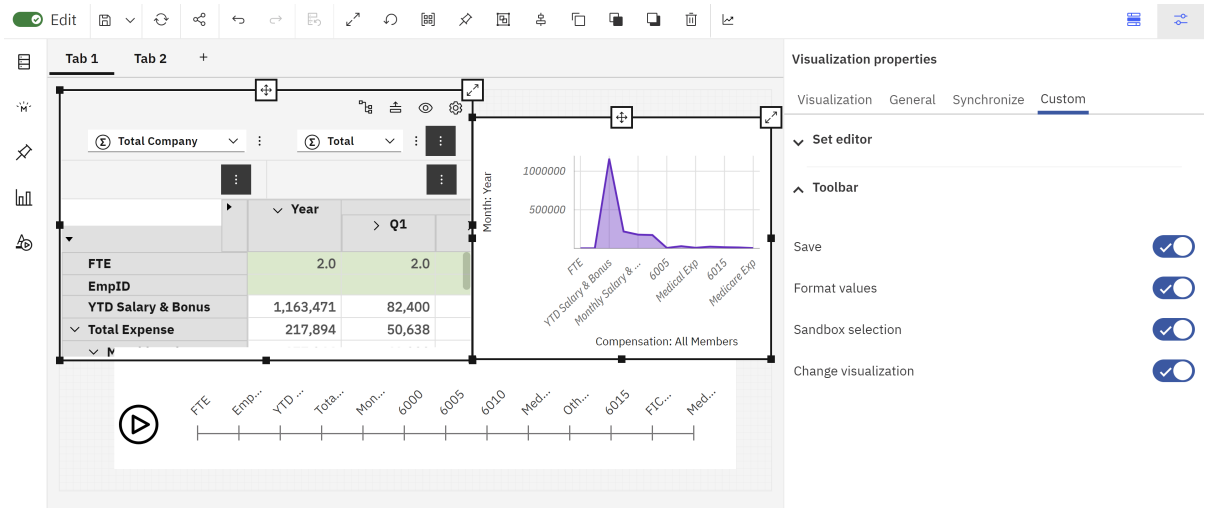
- **Export**
- **Snap commands**
- **Save**
- **Format values**
- **Edit MDX**
- **Sandbox selection**

- **Change visualization**
- **Show/hide overview**
- **Forecast**

For visualizations, you can manage only the **Save**, **Format values**, **Sandbox selection**, and **Change visualization** toolbar icons.

For all other visualizations such as selector widgets, only the **Change visualization** toolbar feature is available to toggle.

If you select both a cube viewer and a visualization in a book, only the icons common to both are available in the **Toolbar** properties.




The settings that you apply in the **Toolbar** properties are retained even if you switch a cube viewer to a visualization or a visualization to a cube viewer. Similarly, the settings are retained if you switch from one type of visualization to another.

# What's new in plans and applications - 2.0.91

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.91.

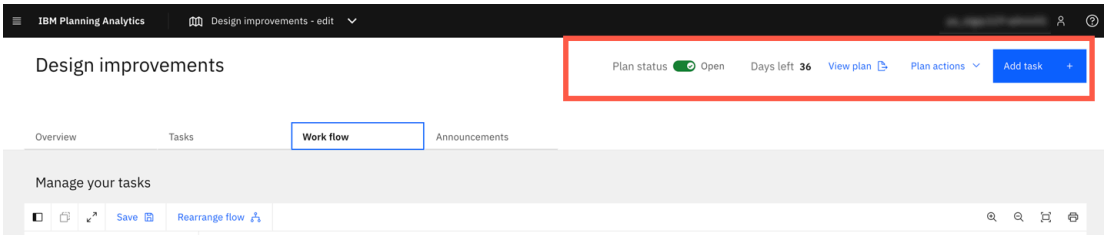
## General design improvements in Applications and Plans

To improve user experience, the Applications and Plans user interface features several design updates in this release.

### Applications and Plans header and menu option updates

In both **Applications** and **Plans**, to improve space utilization, the header and menu options were moved up.

The **Add task** option in **Plans** and **Add a section** option in **Applications** have also moved up to be inline with the other menu options.

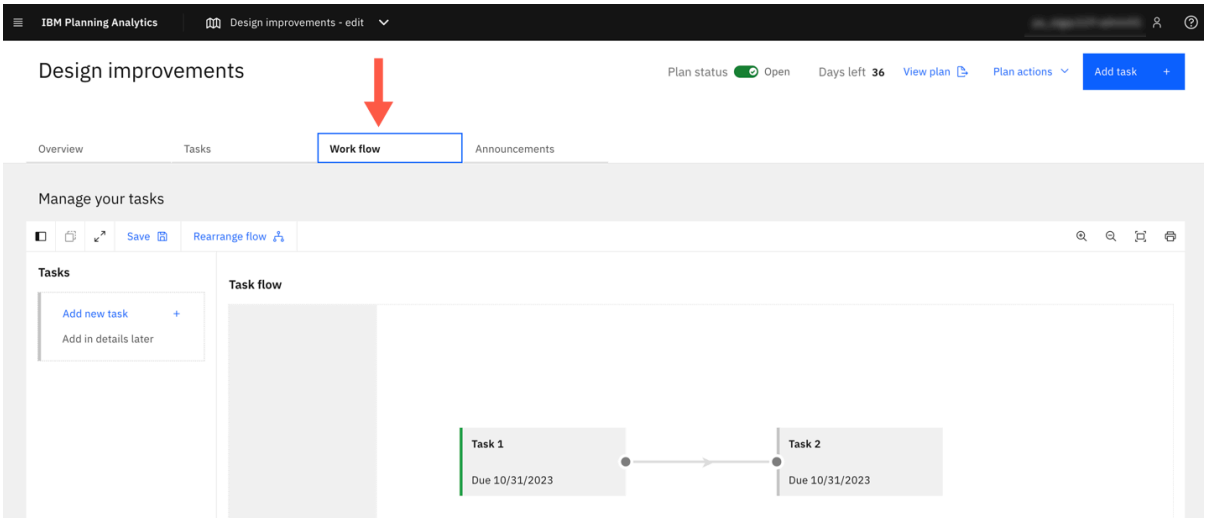



If you have an image in an application or plan header, the image is also now inline with the application or plan name.

If a plan has a specified timeline, the number of days that are left to complete all tasks in the plan now features in the plan header.

### Work flow tab in Plans

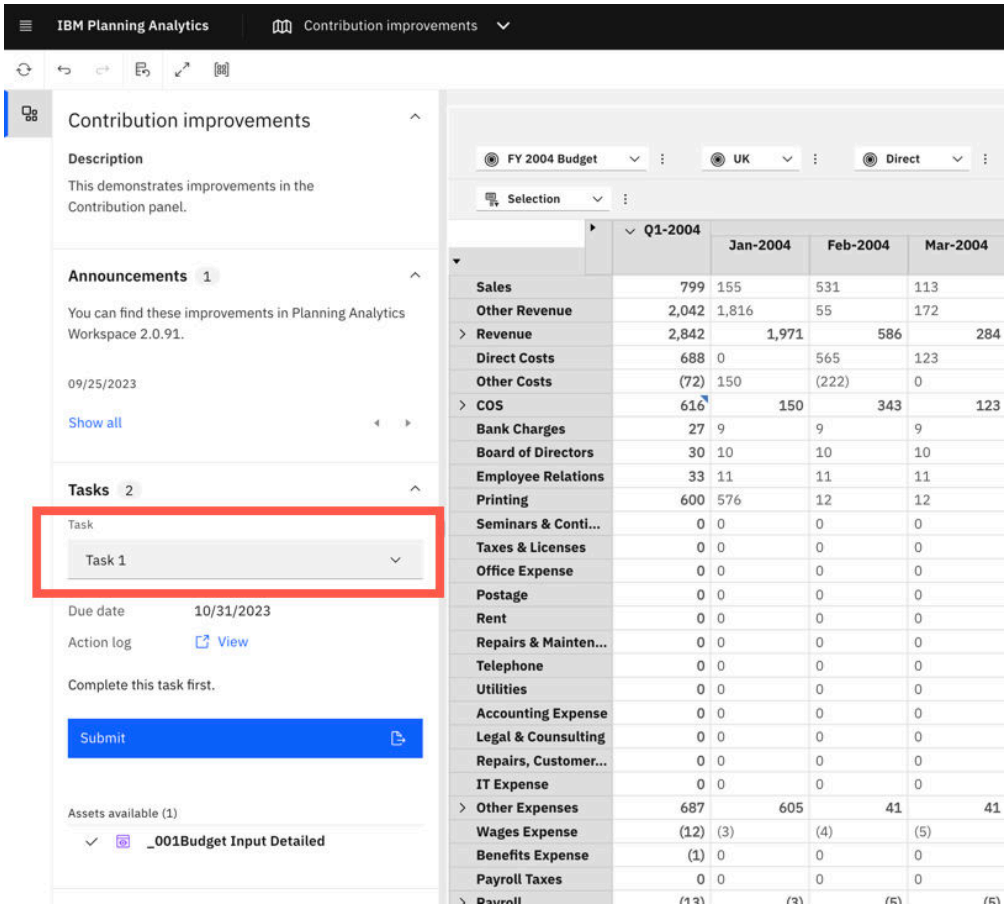
In **Plans**, the flow view was moved to a separate tab called **Work flow**. Previously, you had to switch from the task list view to flow view to view or define task dependencies. Now, you can see the tasks in list view under the **Tasks** tab and the task flow in the **Work flow** tab. You can still create tasks and view and define task dependencies in the **Work flow** tab.




## Usability improvements made to the contribution panel in Plans

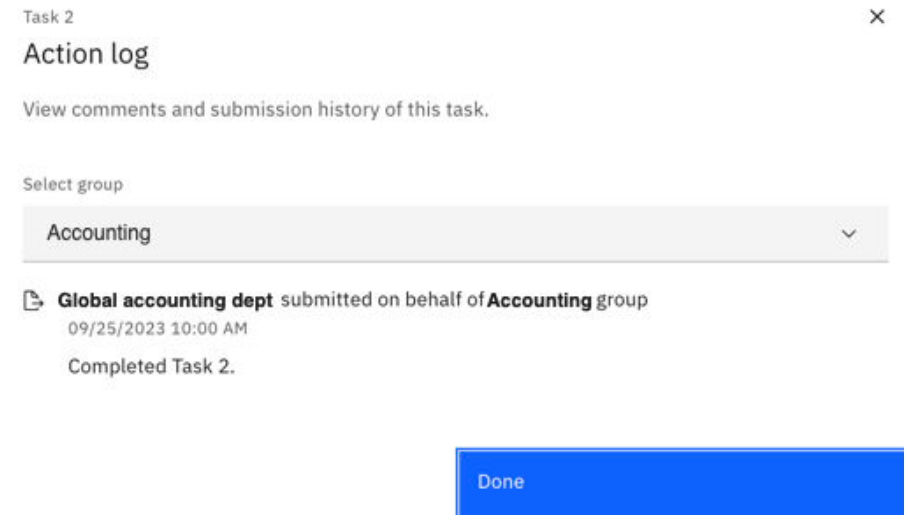
Usability improvements made to the contribution panel in **Plans** now make it easy to view plan information and all the tasks in a plan. These improvements aim to better organize tasks for submissions and approvals, which are especially helpful when there are many tasks in a plan.

You can now access all the tasks from the **Task** list in the **Tasks** section. When you make a selection in the **Task** list, the **Tasks** section updates with the selected task's details such as the due date, action log, and assets.

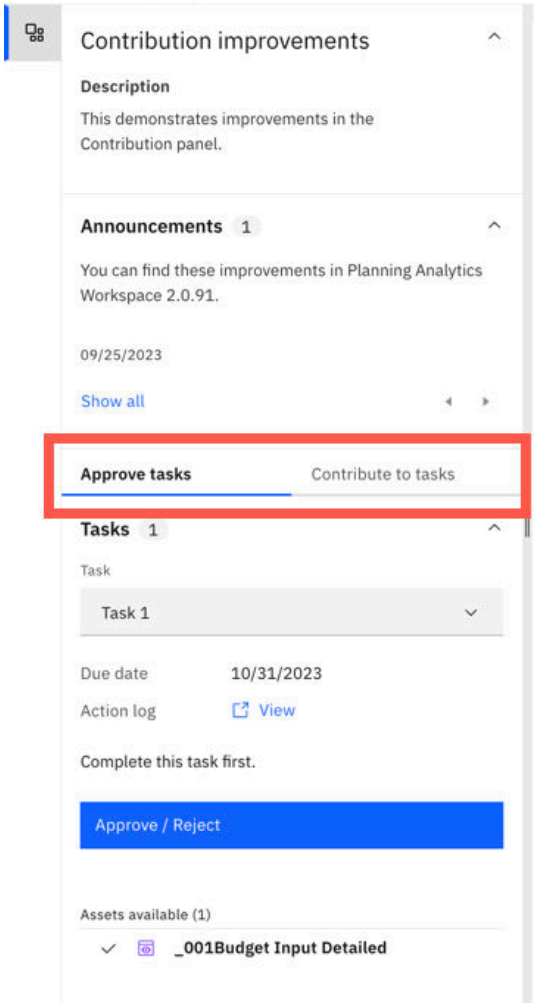



To submit a task, simply select a task from the **Task** list and click **Submit**. When you submit a task, that task no longer displays in the **Task** list.

Click **View** next to **Action log** to see comments and submissions for a selected task.




If you are a Contributor and an Approver for tasks, they continue to display under separate tabs in the panel. These tabs were renamed to **Approve tasks** and **Contribute to tasks**.




As an Approver, select the task you want to approve from the **Task** list and click **Approve/Reject**. After you approve a submission, that task no longer displays in the contribution panel.

For more information, see Plans.

# What's new in modeling - 2.0.91

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.91.

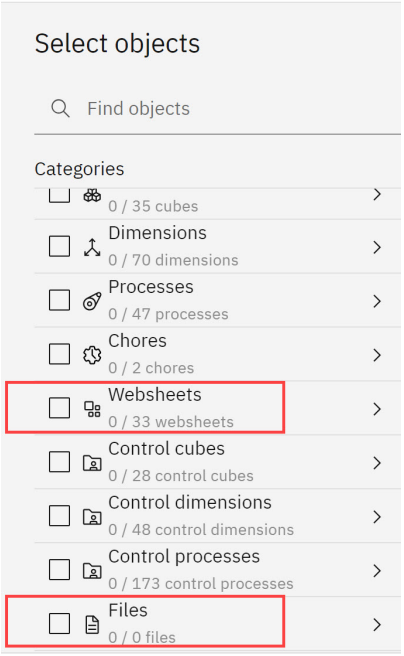
## Extended capabilities in Git repository integration

Git repository integration in Planning Analytics Workspace 2.0.91 introduces support for new objects and files, TurboIntegrator process integration, and the ability to save and reuse a configuration for adding content to a repository.

For full details on Git repository integration, see Use Git repository integration to manage Planning Analytics database objects.

## Support for websheets and files

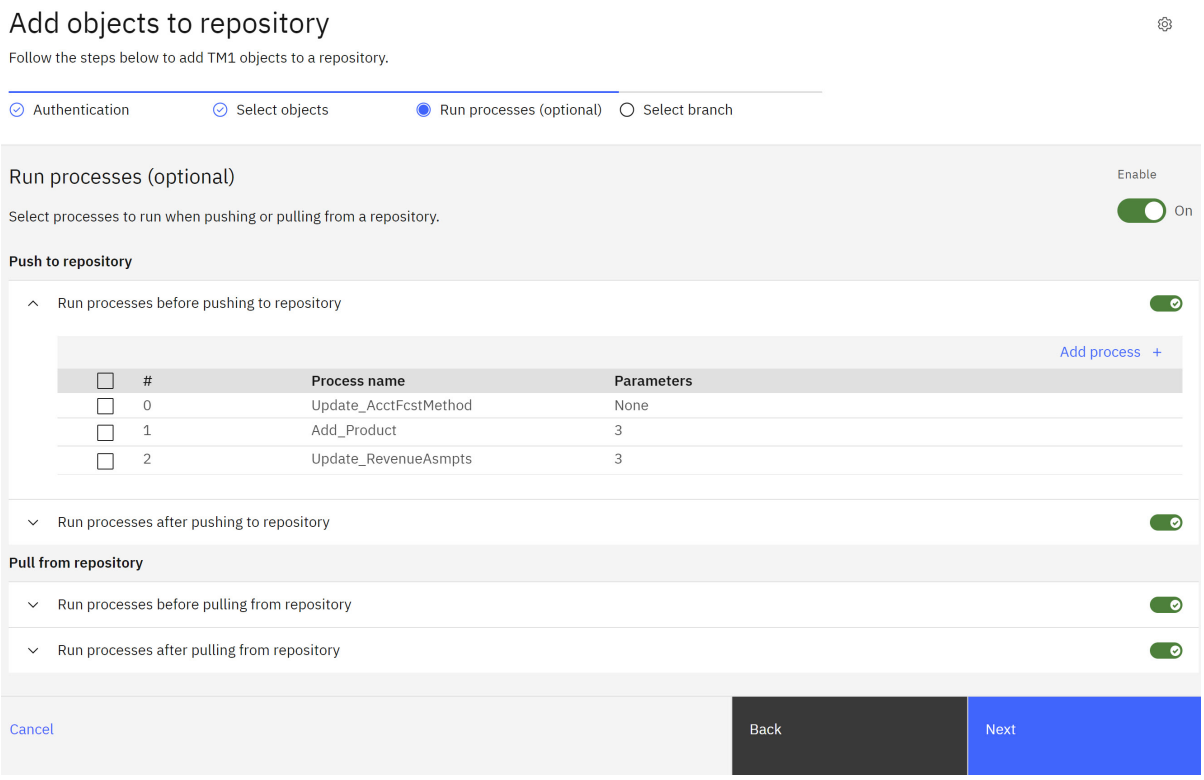
You can now add websheets, as well as any type of file, to your repository.




## Run TurboIntegrator processes in association with a Git push or pull

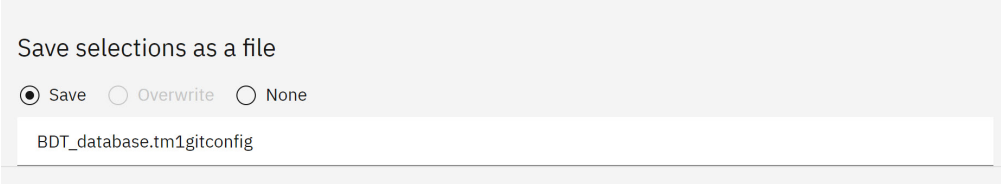
You can run one or more TurboIntegrator processes when you add objects to a repository. You can run processes at any or all of the following points:

- Before pushing to a repository
- After pushing to a repository
- Before pulling from a repository
- After pulling from a repository



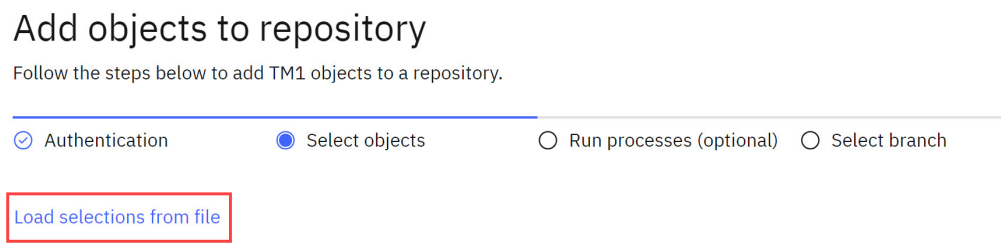
### Save a configuration for adding content to a repository

After you define the objects, processes, and branch to add content to a Git repository, you can save your selections as a file that can be used to quickly set the configuration for later repository updates.




The file is saved to Planning Analytics Workspace File manager with a `.tm1gitconfig` extension.

Upon any subsequent repository updates, you can click **Load selections from file** to apply the selections that are saved in the `.tm1gitconfig` configuration file.




You can modify the repository update configuration after the selections are loaded from the `.tm1gitconfig` configuration file.

### Exclude views when adding cubes to a repository

By default, all associated views are included when you add a cube to a repository. If you want to exclude views, click the **Settings** icon and enable the new **Exclude cube views** option on the **Settings** menu.

## What's new in administration - 2.0.91

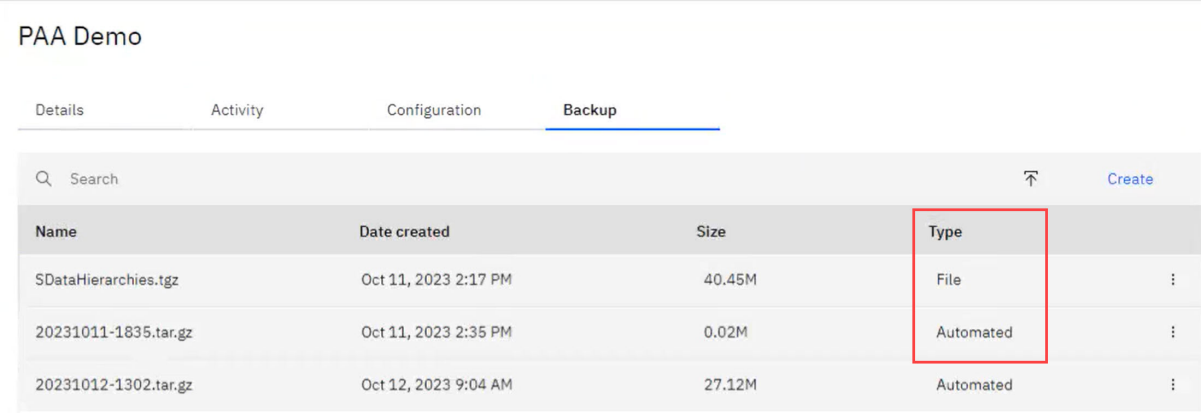
The following new and changed administration features are available in Planning Analytics Workspace 2.0.91.

### Changes to database backup management (Planning Analytics as a Service only)

Planning Analytics Administration 2.0.91 introduces several database administration improvements that are specific to Planning Analytics as a Service.

To work with backups for an individual database, click the database name on the **Databases** page, then click the **Backup** tab.

- You can now view and manage both **File** (user-uploaded) and **Automated** backups for a database on Planning Analytics as a Service.




- When a database is stopped, you can now view the automated backups that are available for the database. Any file (user-generated) backups are not visible when a database is stopped. You can download automated backups when a database is stopped, but cannot upload, create, or restore a backup.
- You can use the **Action** ⋮ menu for a file backup to download, restore, or delete a backup.
- You can use the **Action** ⋮ menu for an automated backup to download or restore a backup.
- Click the new **Upload** ⤒ icon to upload a file backup for a database.

# 2.0.90 - What's new, September 19, 2023

Learn about new features and known issues in version 2.0.90 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

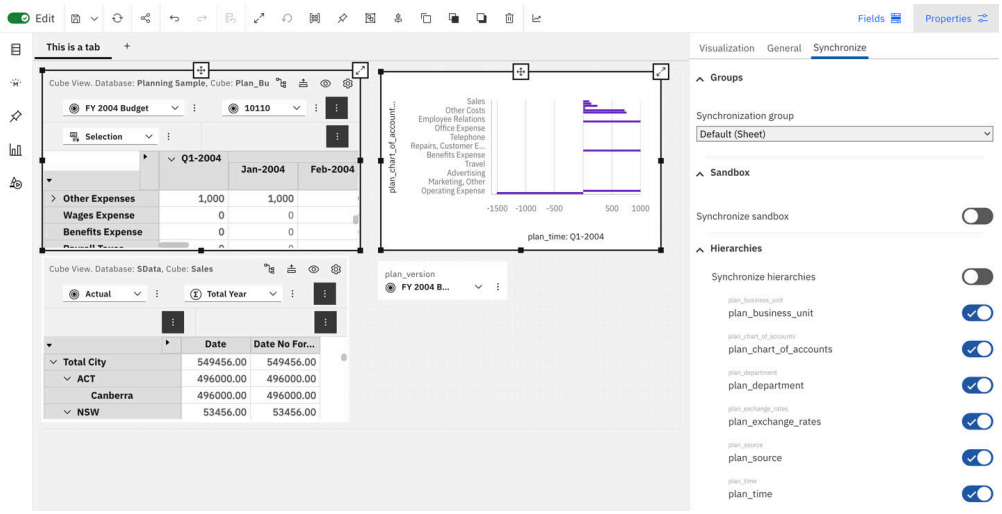
## What's new in books and reporting - 2.0.90

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.90.

### Set synchronization properties for multiple widgets simultaneously

You can now set synchronization properties for multiple widgets at the same time. Previously you could set synchronization properties for only one widget at a time.

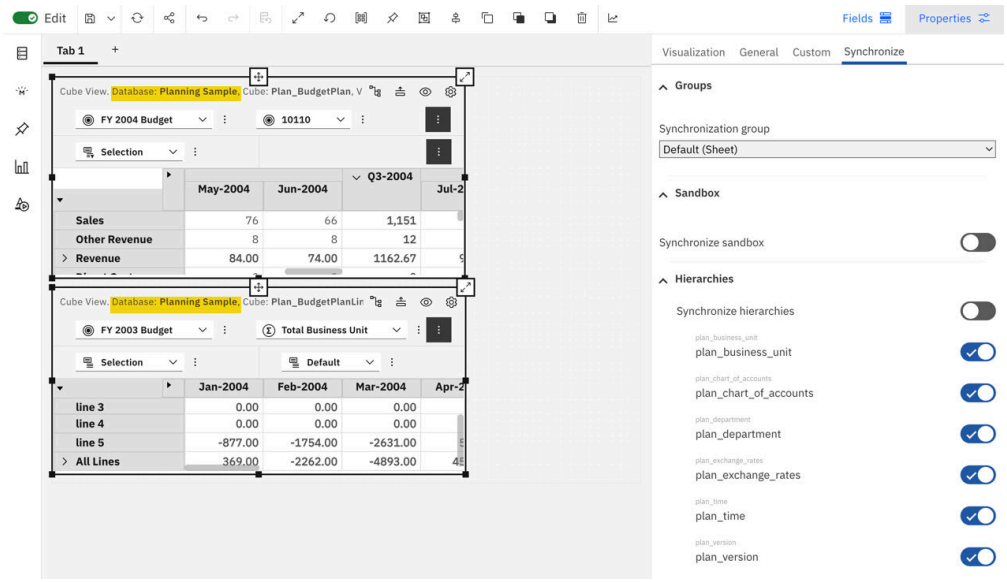
In edit mode, select multiple widgets with Ctrl + click or by using the mouse drag action. Then, under **Properties > Synchronization**, choose the synchronization properties that you want to apply to the selected widgets.




When you select multiple widgets, only the synchronization properties that apply to all selected widgets display. The following properties are available if you select:

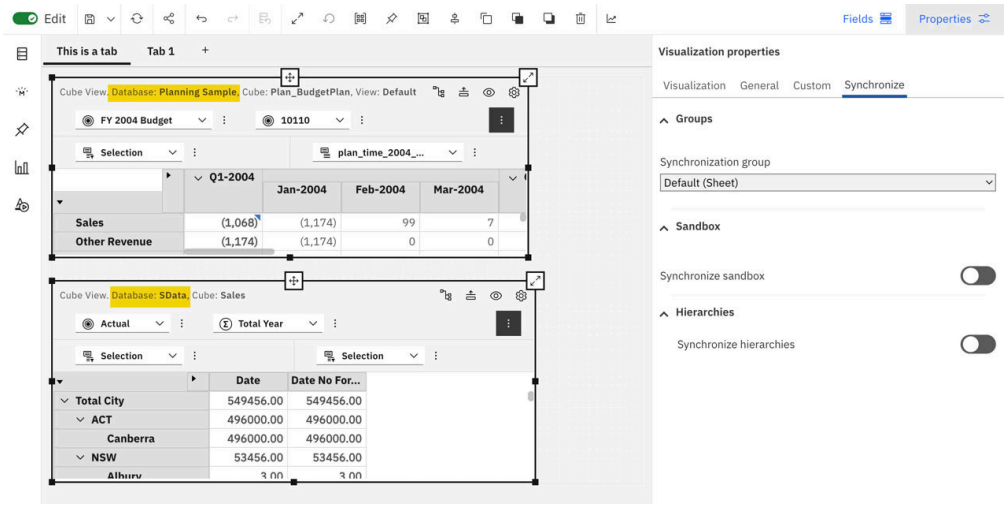
- Cube views from the same database

  The **Synchronization group**, **Synchronize sandbox**, and **Synchronize hierarchies** properties are available. You can synchronize only those hierarchies that are common to both cube views; other hierarchies do not display.



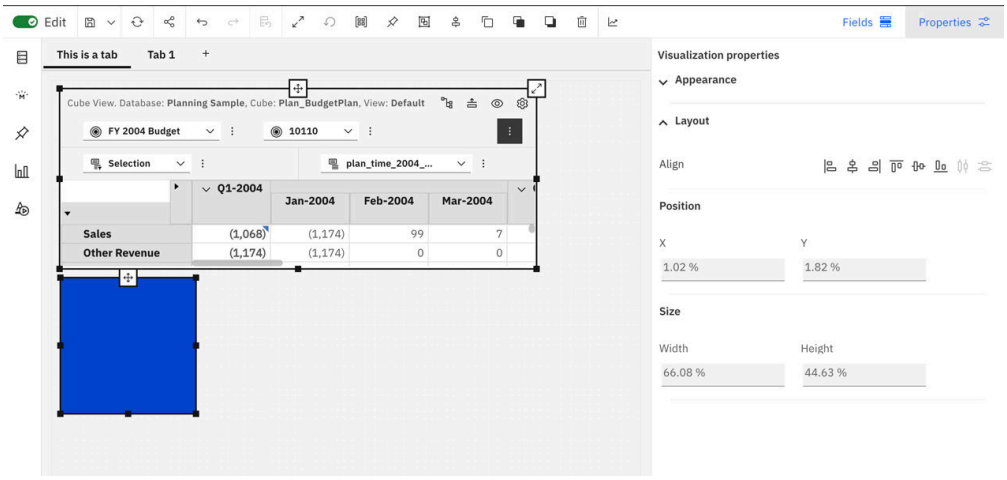

- Cube views from different databases

  The **Synchronization group** and **Synchronize sandbox** properties are available. You can also enable the **Synchronize hierarchies** property to turn on synchronization, but since there aren't any common hierarchies between the selected widgets, there are no hierarchies to select.




- Cube view and shape widget

  Since the shape widget does not have synchronization properties, no synchronization properties display.




The order in which the synchronization properties display in **Synchronization groups** and **Synchronize hierarchies** is determined by the widget that you select first. For example, if the first widget that you select is in the Book synchronization group, the Book option appears first in the **Synchronization groups** list.

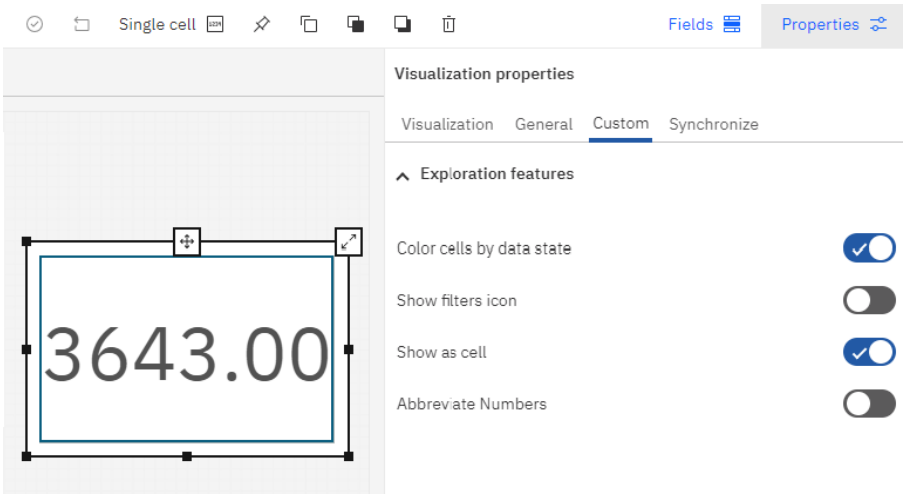
## Ability to delete data from single cell widgets removed

You can no longer use the **Delete** key to delete data from single cell widgets. The **Delete** key function was disabled on the contents of a single cell widget to prevent accidental deletion of data. However, you can modify data in a single cell widget, including clearing the value to display **0**, or delete the single cell widget altogether.

To modify single cell widget values (in consumption and Edit mode), the **Show as cell** property must be enabled for the widget.

To turn on the **Show as cell** property:

1. Select the single cell widget in Edit mode.
2. Go to **Properties** and under the **Custom** tab, enable the **Show as cell** property.




A blue box outlines the value inside the single cell widget. Double-click the value to make changes.

When **Show as cell** property is on, you can also clear the value in a single cell widget with the **Clear** option in the right-click menu. The existing value clears and displays **0** instead.

When the **Show as cell** property is turned off, the single cell widget is in read-only mode. You cannot modify the widget's values or delete the widget.

You can delete a single cell widget in Edit mode only. Use the **Delete** icon in the toolbar to delete the widget.

## Select a member or set by typing its name in an exploration view

The exploration view now offers a quick way to select a member or set. You can open an input box from a dimension tile in the **Context** or **Bench** area to enter a member name or Member ID and update the view with that member. Similarly, you can open the input box from a tile in the **Row** or **Column** area and type the set name to update the view with that set.

This feature is especially useful if you are familiar with the data in the cube and want to avoid using the **Search** feature to find and select a member or set.

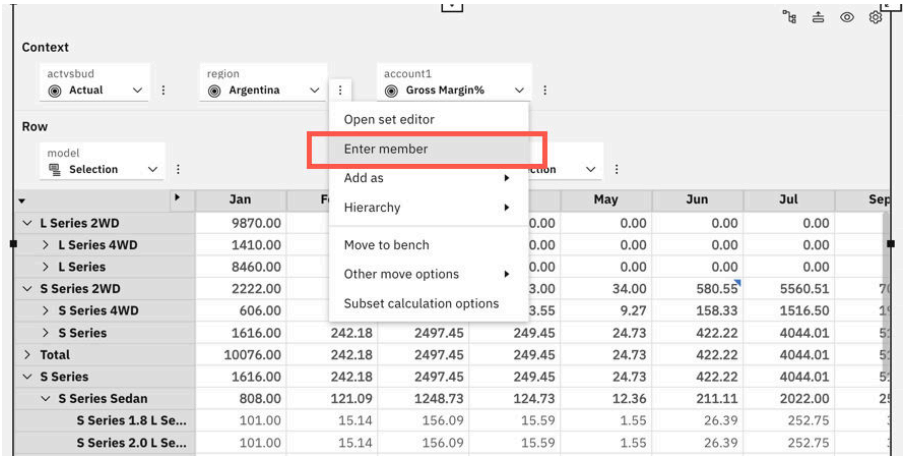
When you enter a member using the input box, the current set is bypassed and you can directly use any member. The set then changes to a single member set.

If you are selecting a set using the input box, the named set you enter must exist. If customized sets are defined, then the named set must be in the customized set list.

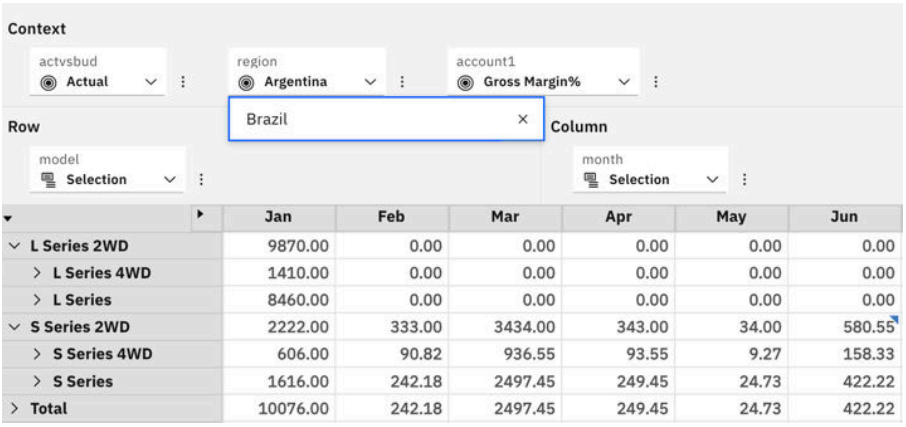
To use the input box to select a member:

1. Click the menu icon next to the dimension tile in the **Context** or **Bench** area and select **Enter member** to open the input box.

   **Tip:** If you're on Windows, simply press **Ctrl + click** the member name in a dimension tile to open the input box.

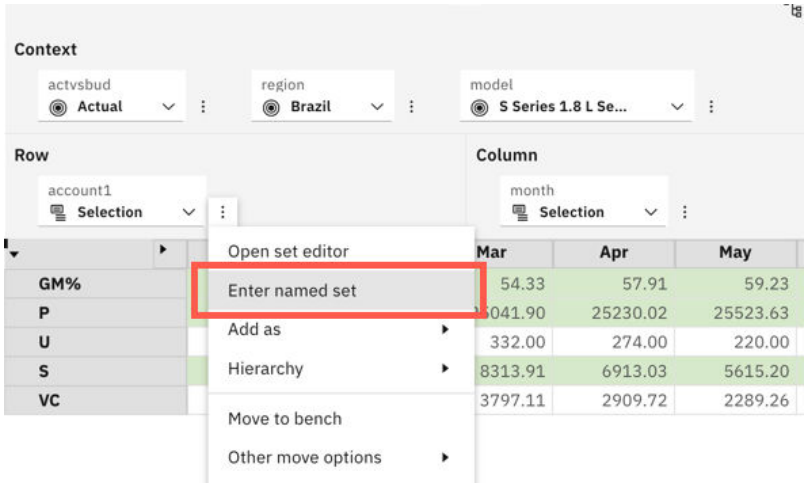



2. Type the member name or ID in the input box and press Enter.



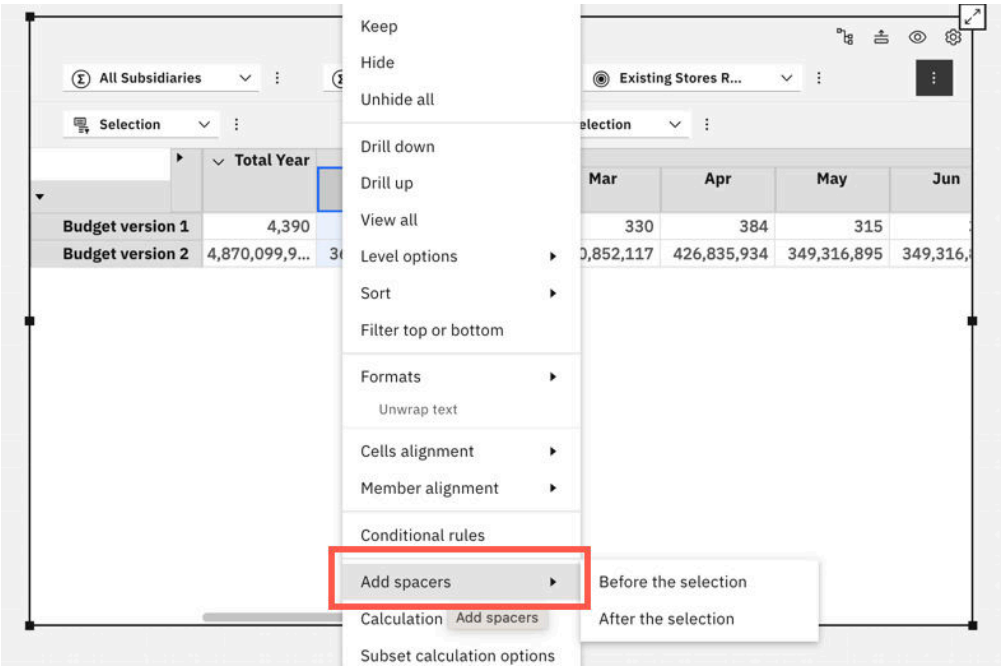

The view updates with the member you entered.

You can select a set by typing its name in the same way from the tiles in the **Row** or **Column** area. If you are using the menu option, select **Enter named set** instead to open the input box.



## Insert spacer rows and columns in explorations

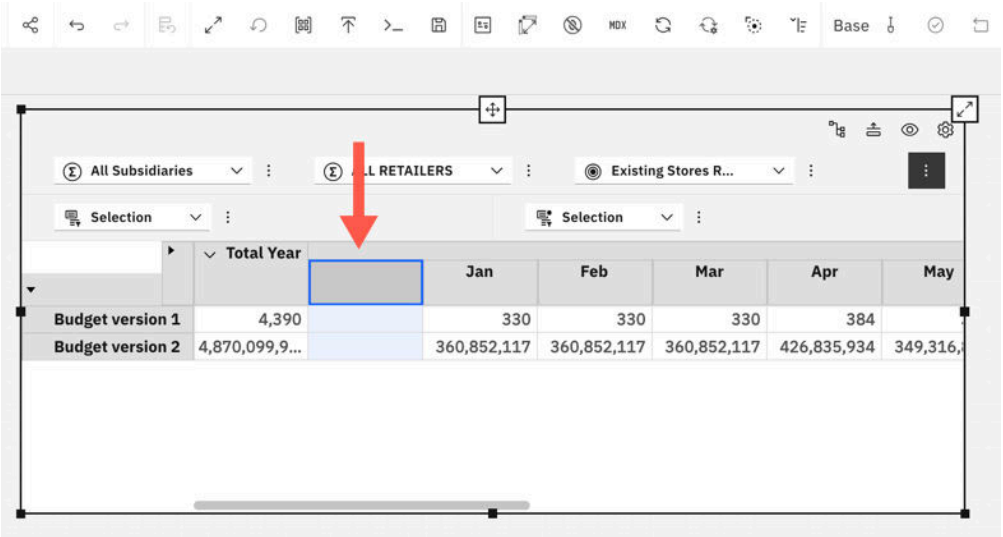
You can now insert spacer rows and columns in explorations through the right-click menu. This is a useful for building stylized views of data in books.




To insert a spacer:

1. Select a row or column and right-click a row or column header in an exploration.
2. Click **Add spacers** and select where you want to insert the spacer row or column.

A spacer row or column is inserted at the position you selected.

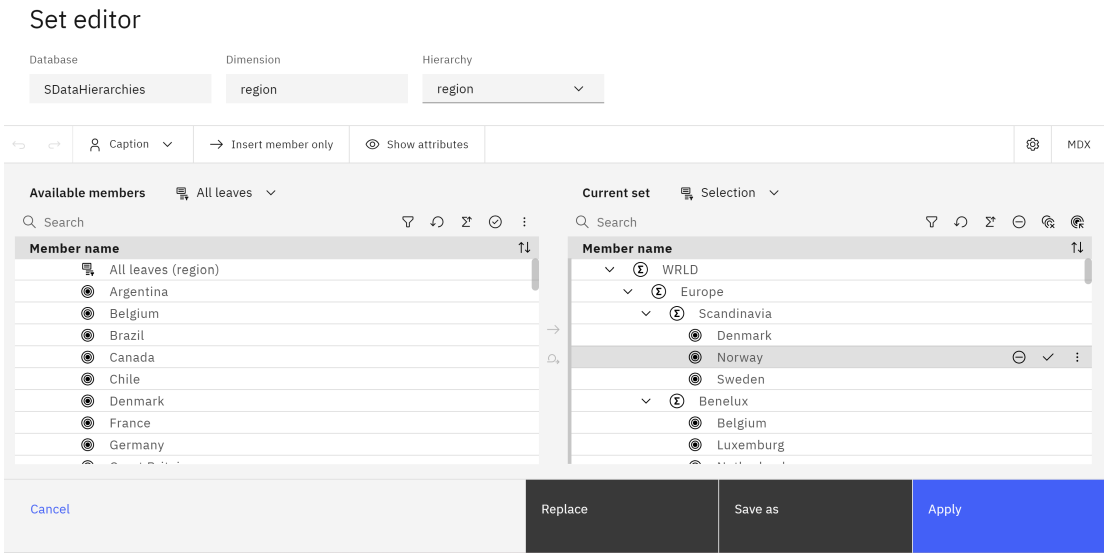



You can copy and paste a row or column with spacer cells. However, the spacer cells can break some spreads.

To remove a spacer row or column, right-click the spacer header cell and select **Remove**.

## Replace button now works on selections in the Current set

The **Replace** button in the set editor now also works on the selections you make in the **Current set** pane. You can select members on both sides of the set editor, either the **Available members** or the **Current set**, and replace members in the view.



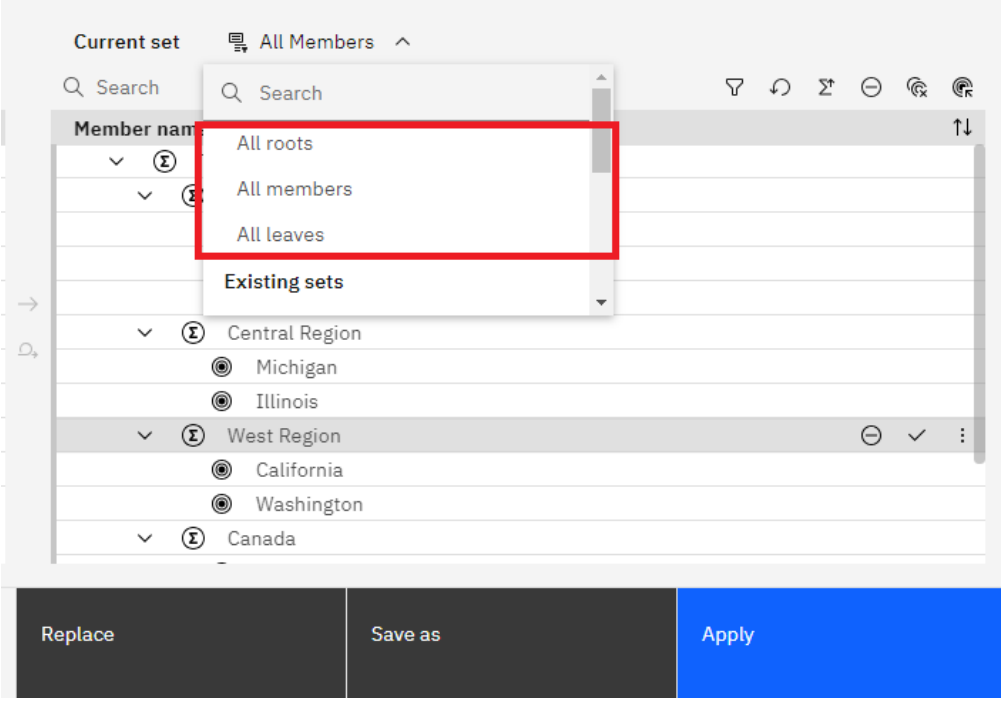

The **Replace** button is disabled when no members are selected in the set editor.

Previously, while you could select members on one or both sides of the set editor at the same time, the **Replace** button only worked on selections that were made in the **Available members** pane. To replace members in a view from the **Current set** in the set editor, you had to click **Apply** instead of **Replace**. Additionally, if there were no selections in the **Available members** pane, the **Replace** button was disabled.

## Virtual set and reset features are now available for the current set in the set editor

All virtual sets (**All roots**, **All members**, and **All leaves**) in the **Available members** pane in the set editor are now also available in the **Current set** pane. Additionally, reset features are now available in the **Current set** pane.

The availability of virtual sets in the **Current set** pane lets you work with these sets without having to toggle **Available members** back on when you are in single-pane mode.




When you select a virtual set, the **Current set** label changes to display the virtual set's name.

The **Current set** changes when you:

- Select a named set or virtual set from **Current set** list
- Replace **Current set** with a named set from the **Available members** pane
- Switch hierarchies - when you switch hierarchies, the **Current set** changes to the default named set. If a default named set does not exist, the **Current set** switches to **All roots**.
- Delete the current named set

If your **Current set** is a named set, the **Current set** keeps the alias of the existing set and expands above with the virtual set, when you select **Show totals trailing** and choose a virtual set.

You can use **Undo** and **Redo** to move between a virtual and a named set.

You can now also **Reset the current set** to reset changes that you made to the currently selected set.

You can reset the current set only if you made edits to the set or if you inserted members from **Available members** to the **Current set**. You can verify that you made an edit by the state of the set icon: .

The **Reset the current set** feature is disabled when you perform the following gestures:

- View the attributes of any member in the **Current set** by clicking **Show attributes**
- Select a different set from **Current set** menu
- Switch the hierarchy in the set editor
- Delete a set on the **Available members** pane
- Save a new named set

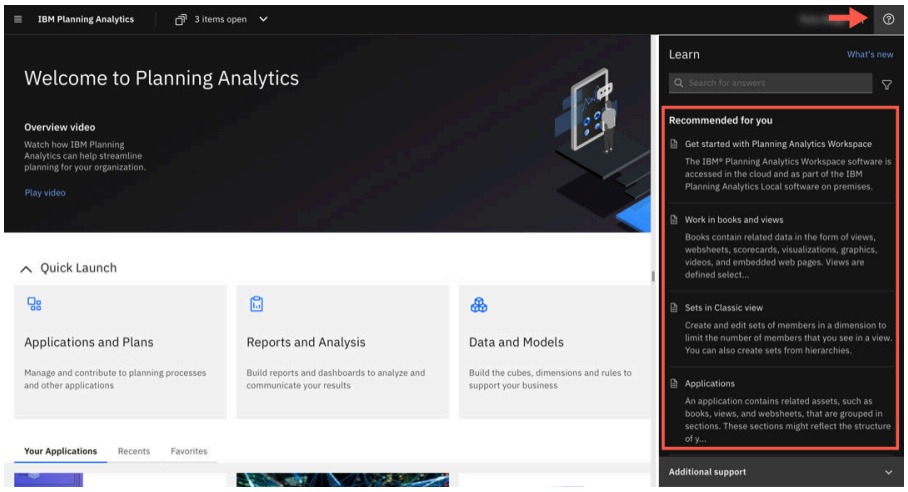
## AI-generated recommendations available in the Learn Pane

The Learn Pane now provides dynamic AI-generated recommendations as well as static recommendations.

When you interact with the content inside the Learn Pane, the recommendation engine makes daily updates based on what you and other users are viewing. If no content was viewed inside the Learn Pane,

the recommendations remain unchanged. For example, if you and other users use books and views more often, more content about books and views is generated in the Learn Pane.

Previously, the Learn Pane provided suggestions only based on where you are in the product. The new update makes it convenient to find helpful topics that are more tailored to your current needs.




# What's new in plans and applications - 2.0.90

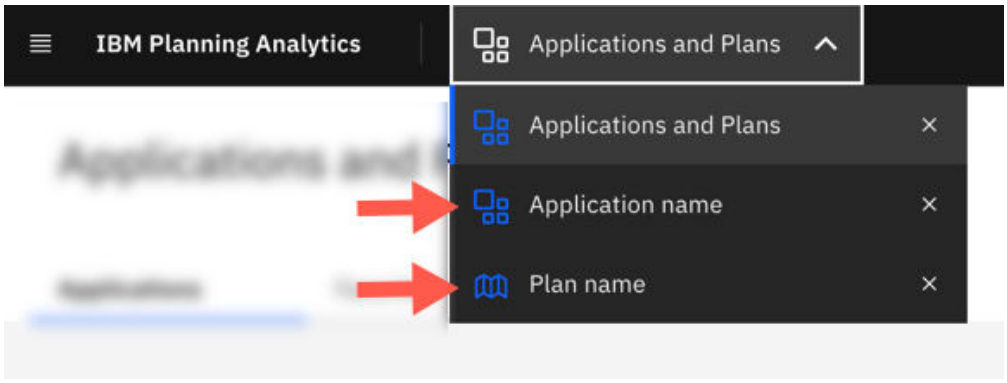
The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.90.

## Usability improvements to the contribution page in Applications and Plans

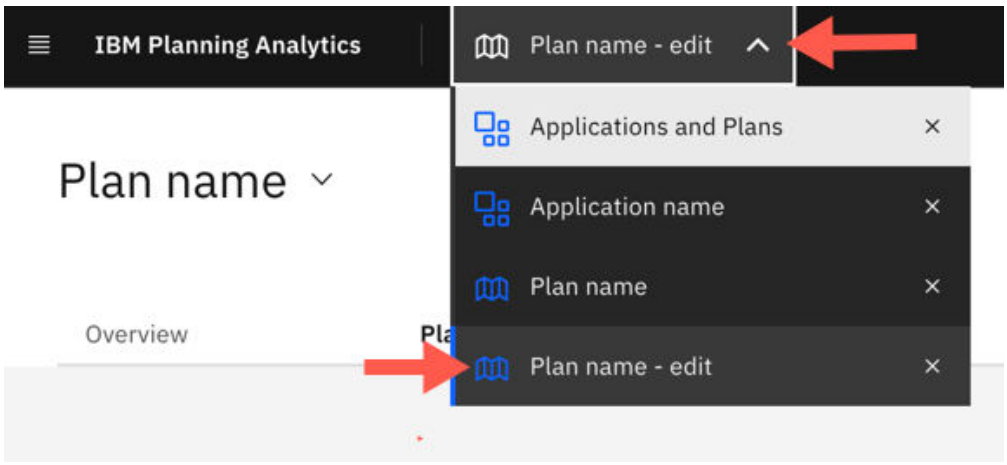
The contribution page in Applications and Plans include several usability enhancements that make it easier to navigate between applications and plans or to work with assets.

### Easy to identify contribution mode in the content menu

When you open an application or a plan to the contribution page, the application or plan name displays in the content menu.




You can now easily differentiate between an application or plan that is in contribution and edit mode. An application or plan that is in edit mode displays in the content menu with an "- edit" appended to its name.




### Assets open in the same page

Assets now open in the same page as the application or plan when you access them on the contribution page. Previously, when you opened an asset in an application or plan, the asset opened on a new page and displayed as a separate page in the content menu.

### Editing of assets restricted in contribution mode

You can no longer edit assets in applications or plans when you are in contribution mode. You can enter values in a cube viewer or change the visualization type in contribution mode but you cannot add a new widget to the asset or change the asset's properties.

You can edit an asset in **Reports and Analysis**.

## What's new in modeling - 2.0.90

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.90.

### Work with database logs on the modeling workbench

You can now access the server log, transaction log, and audit log for a database directly from the modeling workbench in Planning Analytics Workspace.

### About this task

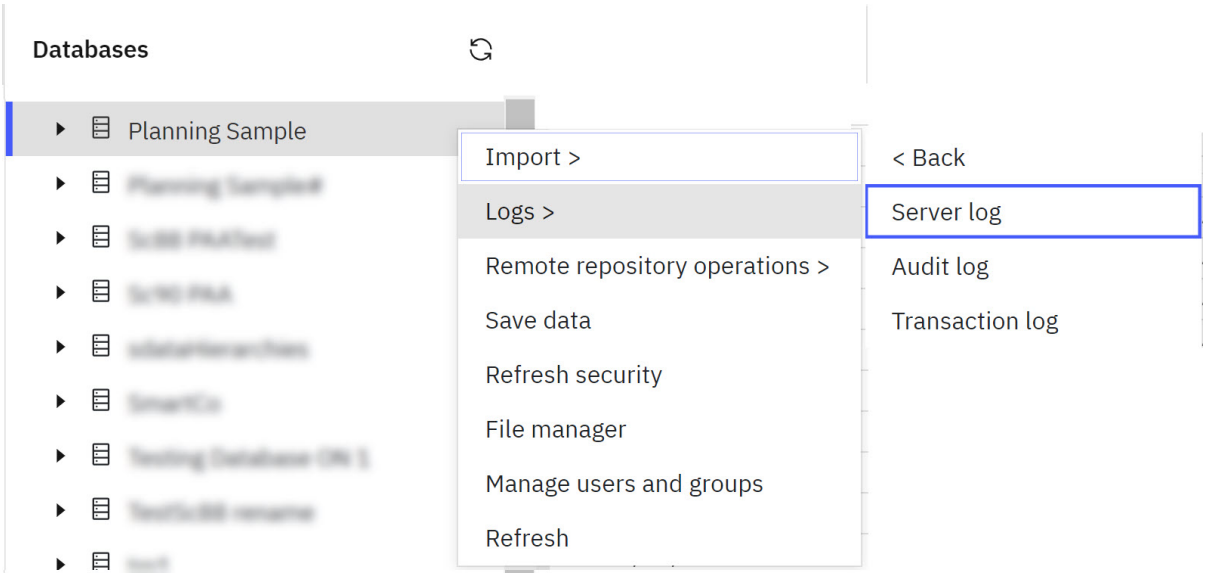
Access to server logs is available only on the modeling workbench. You cannot open logs from a book.

The option to open the server log and the transaction log is available for all databases. To access the audit log, the **Audit log** configuration parameter must be enabled for the database.

You can still download database logs from the Databases page in Planning Analytics Administration.

### Procedure

1. To open any of the available database logs, open a modeling workbench.
2. Right-click the database on the **Data** tree.
3. Click **Logs**.
4. Click the log that you want to open.




## Results

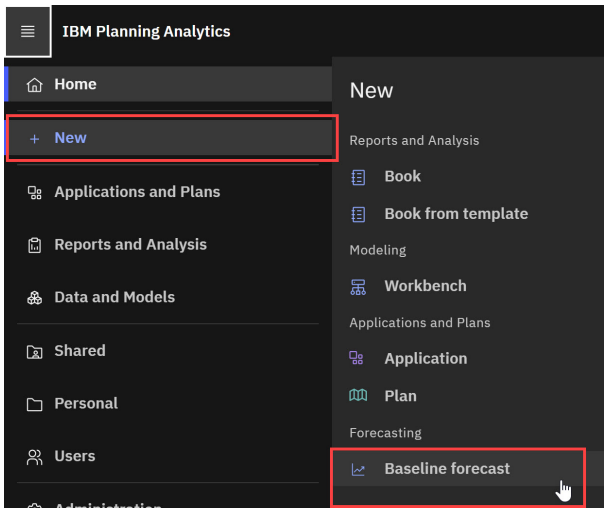
The log you click opens in a new tab on the modeling workbench. All logs include a filter to specify the period to display records. The audit log and transaction log also include additional filters that you can apply to the records shown in the log viewer.

## Use baseline forecasting to create and share forecasts

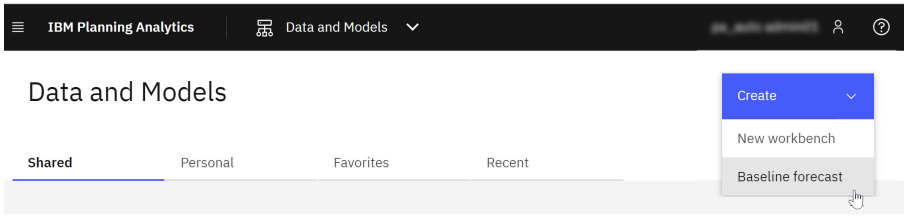
A baseline forecast is a new Planning Analytics Workspace asset that administrators and modelers can create and share with other users in your organization. Administrators, modelers, and analysts with the appropriate permissions can run the baseline forecast at any time and use the forecast to improve the planning process. Consumers cannot run baseline forecasts.

Baseline forecasts enhance the existing forecasting capabilities in Planning Analytics Workspace. You can still create and run ad-hoc forecasts on explorations in a book.

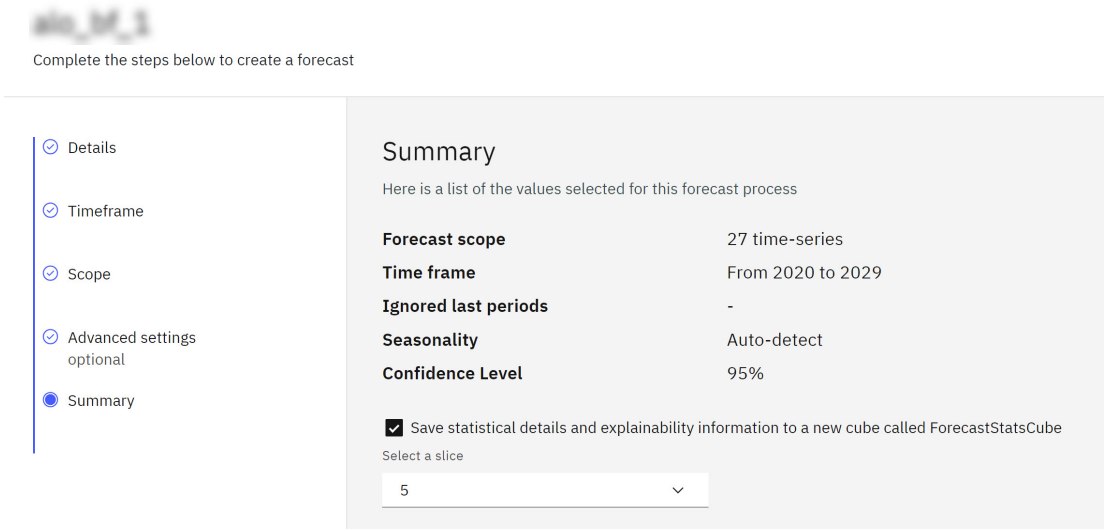
You can create a baseline forecast from the Planning Analytics Workspace home menu.



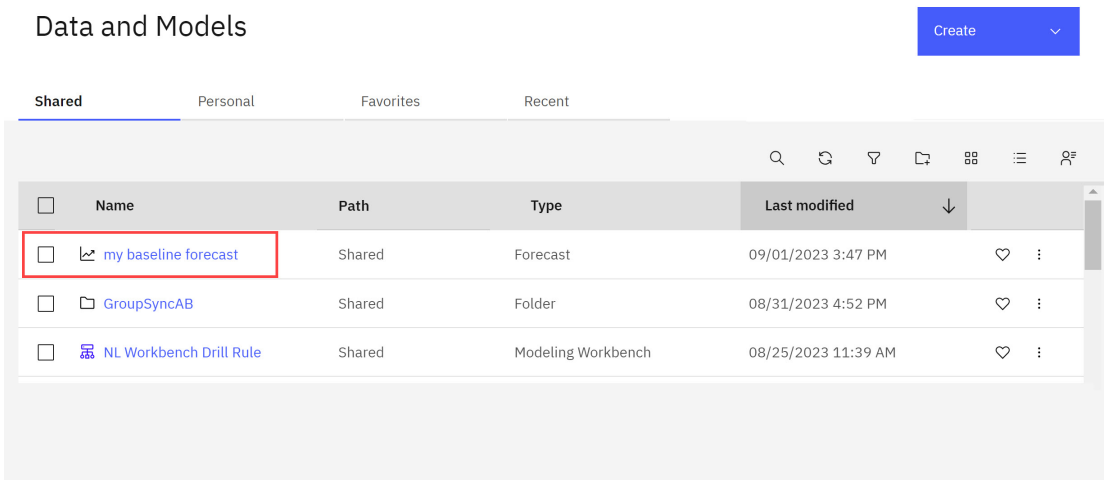

You can also create a baseline forecast from the Data and Models page.




Baseline forecasts are created using the same settings and parameters you are familiar with from creating ad-hoc forecasts on explorations. You set the timeframe, scope, and optional advanced settings for the forecast.




After the baseline forecast is defined, saved, and activated, it's available as an asset on both the Data and Models page and the Reports and Analysis page, where other users can open and run the forecast.




# What's new in administration - 2.0.90

The following new and changed administration features are available in Planning Analytics Workspace 2.0.90.

## Simplified process to rename databases, snapshots, and user groups

A new **Edit** option on the **Databases**, **Lifecycle Management**, and **Users and groups** pages simplifies the renaming process.

In Planning Analytics Administration, select a database, snapshot, or user group on the relevant administration page, then click Edit to modify the item name. You can also modify the description for snapshots and user groups.

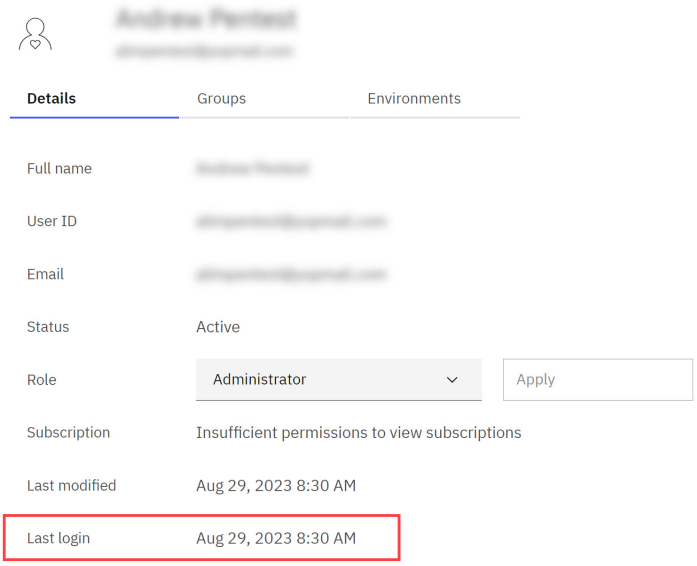
Note that a database must be stopped before you can rename it.

## View a user's last login in Planning Analytics Administration

An enhancement in Planning Analytics Workspace 2.0.87 added the ability to view the last login for a user when you downloaded a user report from the **Users and groups** page. You can now view the last login for a user directly in Planning Analytics Administration without downloading a report.

### Procedure

1. Open Planning Analytics Administration.
2. Click the **Users and groups** tile.
3. Click the **Users** tab.
4. Click a user.
5. Click the **Details** tab for the user to review the last login.




# Deprecation notices - 2.0.90

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.90.

## Classic cube viewer removed from Planning Analytics Workspace 2.0.90

As of Planning Analytics Workspace 2.0.90, the classic cube viewer is no longer available in Planning Analytics Workspace.

Any existing classic cube viewer will be converted to a new experience cube viewer when first opened in Planning Analytics Workspace 2.0.90.

# 2.0.89 - What's new, July 24, 2023

Learn about new features and known issues in version 2.0.89 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

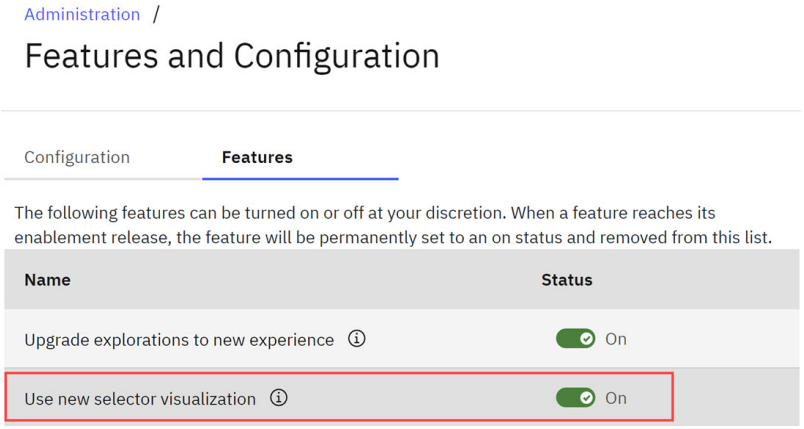
## What's new in books and reporting - 2.0.89

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.89.

### Use the new Selector visualization in place of selector widgets in your books

In Planning Analytics Workspace 2.0.89, a new **Selector** visualization is available to replace selector list and selector tile widgets in books.

**Important:** The new **Selector** visualization is partially subject to enablement by a Planning Analytics Workspace administrator.

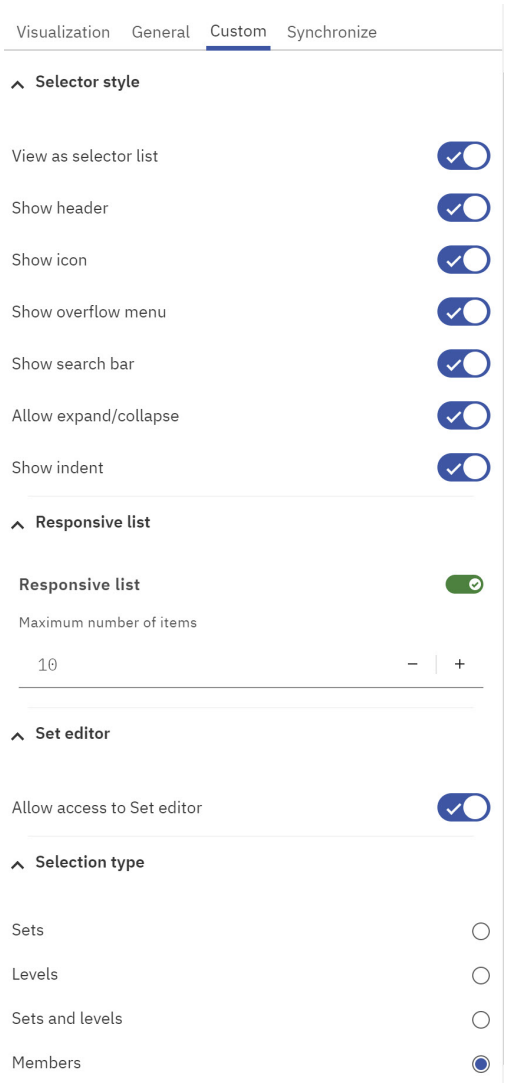



If an administrator does not explicitly enable the new **Selector** visualization feature, existing selector list and selector tile widgets in books remain unchanged until the enablement release for this feature, which is scheduled for Planning Analytics Workspace 2.0.93.

When the new Selector visualization feature is enabled, any existing selector list or selector tile widget in a book is converted to a Selector visualization when you open the book. The visualization is styled to most closely replicate the widget it replaces.

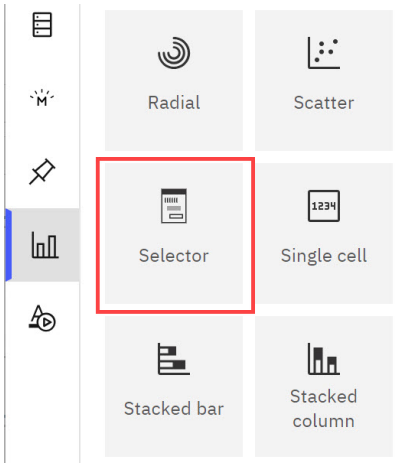
Regardless of the enablement status of the new Selector visualization feature, Selector visualizations can be created from the Visualization pane in Planning Analytics Workspace 2.0.89.

The Selector visualization behaves much as selector list and selector tile widgets did, but there are multiple **Custom** visualization properties that you can apply to tailor the visualization for your purposes.




There are several ways to insert a Selector visualization into a book. In all cases, the book must be in Edit mode.

- From the **Data** tree:
  1. On the **Data** tree, fully expand the **Dimensions** node beneath the cube for which you want to create a Selector visualization. You can create a Selector only from the Dimensions node beneath a cube. You can't create a Selector from the main Dimensions node.
  2. Right-click the dimension, hierarchy, or subset that you want to use as a Selector.
  3. Click **Add as selector**.
- From the **Visualizations** pane:
  1. Click the **Visualizations** icon to open the pane.
  2. Click the **Selector** icon.



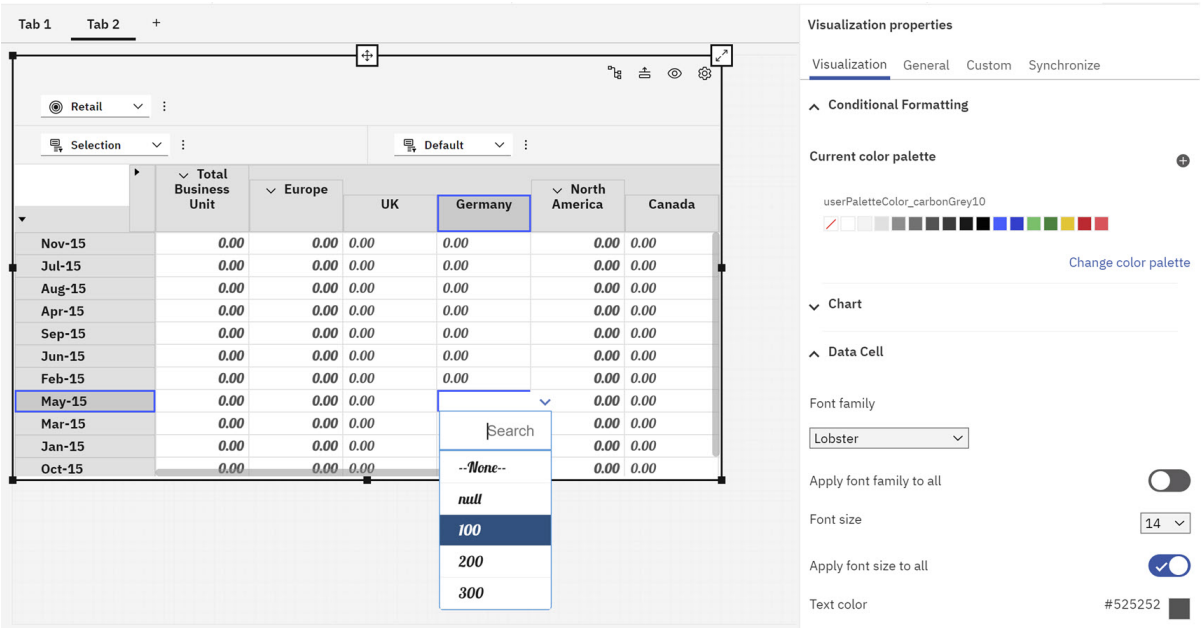

3. From the **Data** tree, drag and drop a dimension, hierarchy, set, or level onto the visualization placeholder.

- From an exploration, click the dimension tile that you want to use as a selector, then drag the dimension to an empty space on the book.

## Pick lists now use the font defined for data cells in an exploration

In previous versions of Planning Analytics Workspace, pick list cell values and list members were displayed using the default IBM Plex® font.

As of Planning Analytics Workspace 2.0.89, pick lists cells and pick list members use the font defined for the exploration by the **Data cell** font family property.




This example shows the exploration, including pick list cells and the pick list members, using the Lobster font.

## Resize books to fill space in a tab

You can now resize a book in a tab that contains the full width and height within different window sizes with the new **Fill space** feature.

Previously, the aspect ratio of the book and its contents was maintained. Now, all the contents in a book fit perfectly within the browser window and the aspect ratio setting is ignored. You can apply the **Fill**

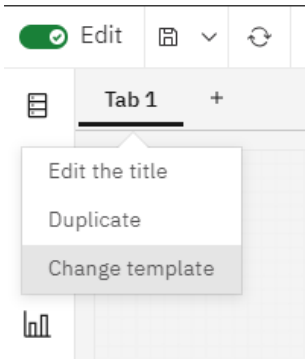
**space** feature in several places: when creating a new book with tabs, a new book from a **Fill space** template, and in **Properties**.

**Note:** When you save **Views** or **Websheets**, they are saved as a **Fill space** layout by default.

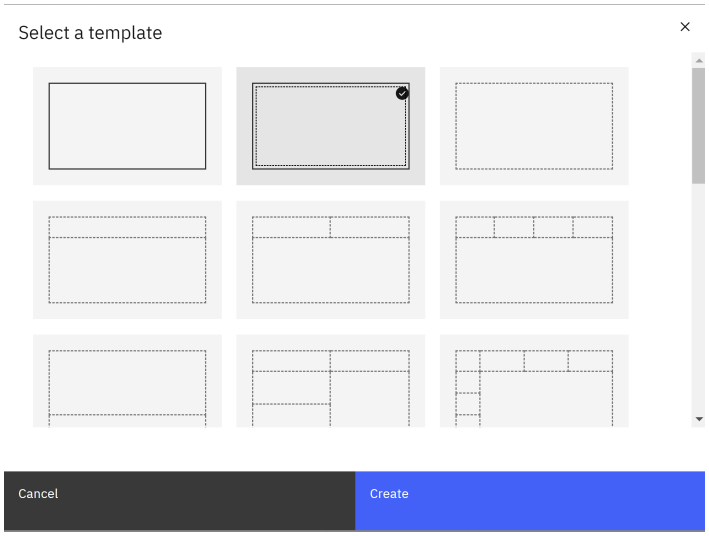
## Fill space in book tabs

You can apply the **Fill space** feature when creating a new book.

1. From the **Home** menu, click **+ New** > **Book**.
2. In **Edit** mode, click **Tab 1** then click **Change template**.




   a. Click the second template in the first row, then click **Create** to change the layout to **Fill space**.




3. Alternatively, you can click the **Add new tab** icon **+**.
   a. Click the second template in the first row, then click **Create** to change the layout to **Fill space**

## Fill space in book from template layout

You can use the **Fill space** feature when creating a **Book from template**.

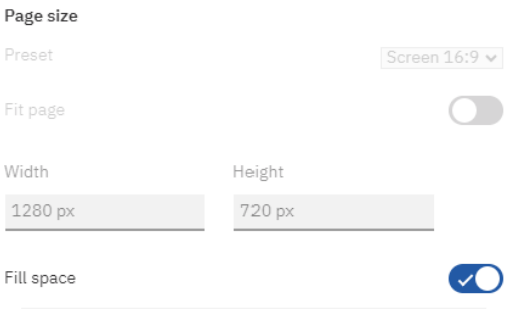
1. From the **Home** menu, click **+ New** > **Book from template**.
2. Click **Fill space** and select the template, then click **Create**.

## Fill space in properties

You can set the **Fill space** feature in the **Properties** panel on widgets in the current tab of a book.

1. Click **Properties**.
2. Under **General**, click **Canvas** and go to the **Page size** section.

3. Set the **Fill space** toggle to enable the feature.



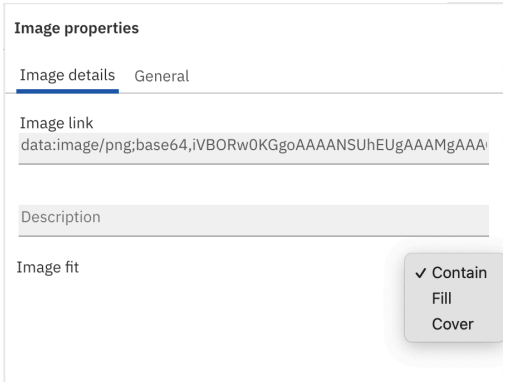

### Image fit

For image widgets, you can also apply the **Image fit** property that's found in the **Properties** panel by choosing any of the following options:

- **Contain** – the default option for an image widget that locks the image's width and height when you resize the image.
- **Fill** – fills up the image in the entire widget container but stretches the image when you resize it.
- **Cover** – maintains the height of the image and crops the width when you adjust the size.

To add images to a book see Add pictures, media, and web pages.

You can resize your image widgets by clicking the image, then under **Image details** click the **Image fit** drop-down menu to view all the options.




## What's new in plans and applications - 2.0.89

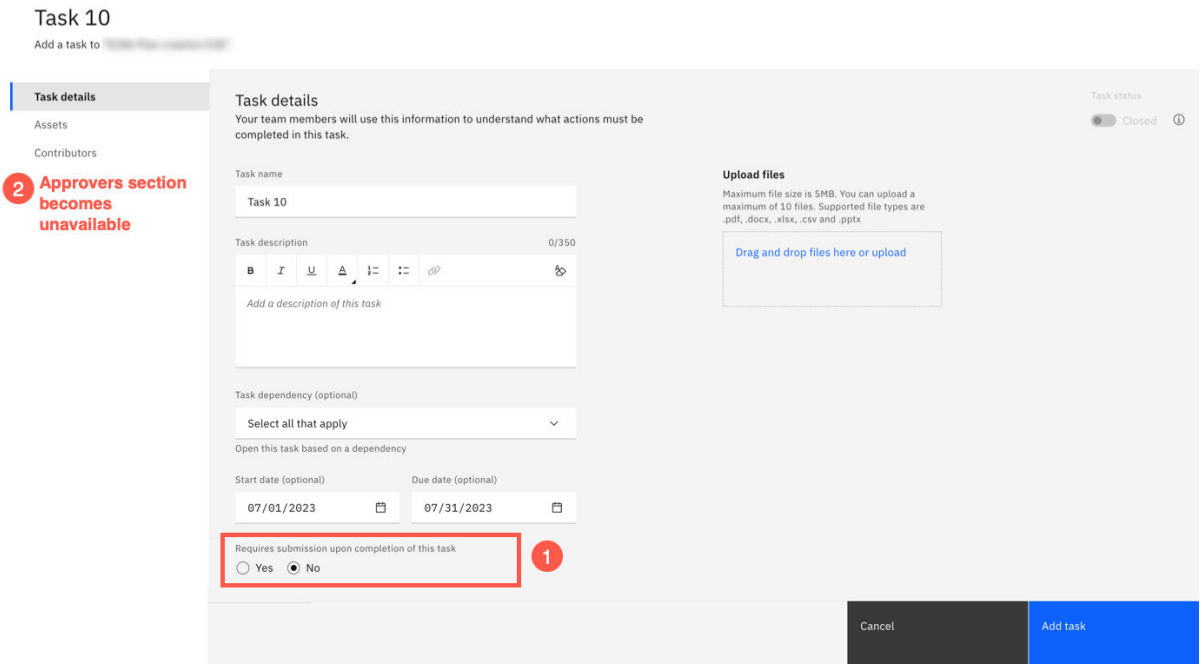
The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.89.

### Complete button for tasks that do not require submission

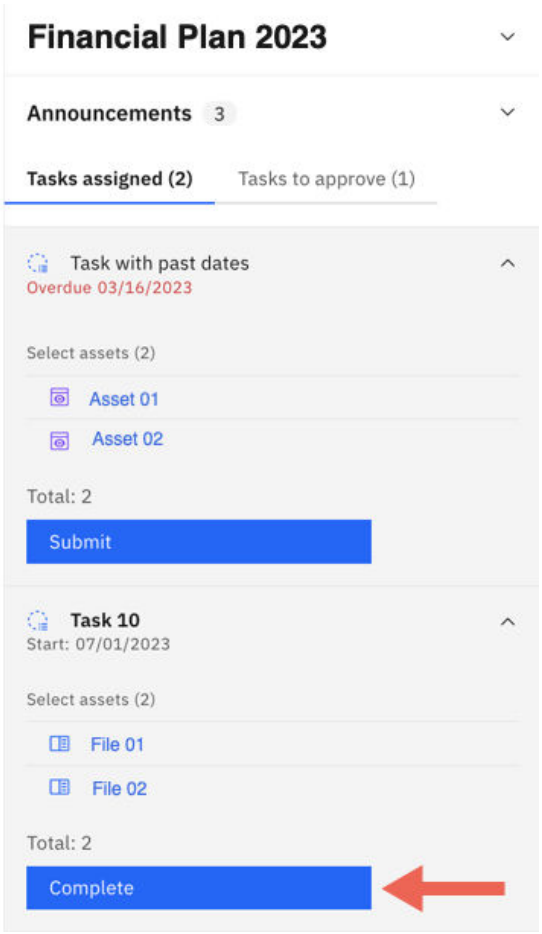
A **Complete** button is now available on the contribution page in tasks that do not require submission. Previously, if a task in a plan did not require submission, there was no way to indicate that the task was complete.

Plan owners and administrators can configure a task to not require a submission. When a submission is not required:

- the **Approvers** section becomes unavailable in the task creation page




- the **Complete** button displays on the plan contribution page for the task




## What's new in modeling - 2.0.89

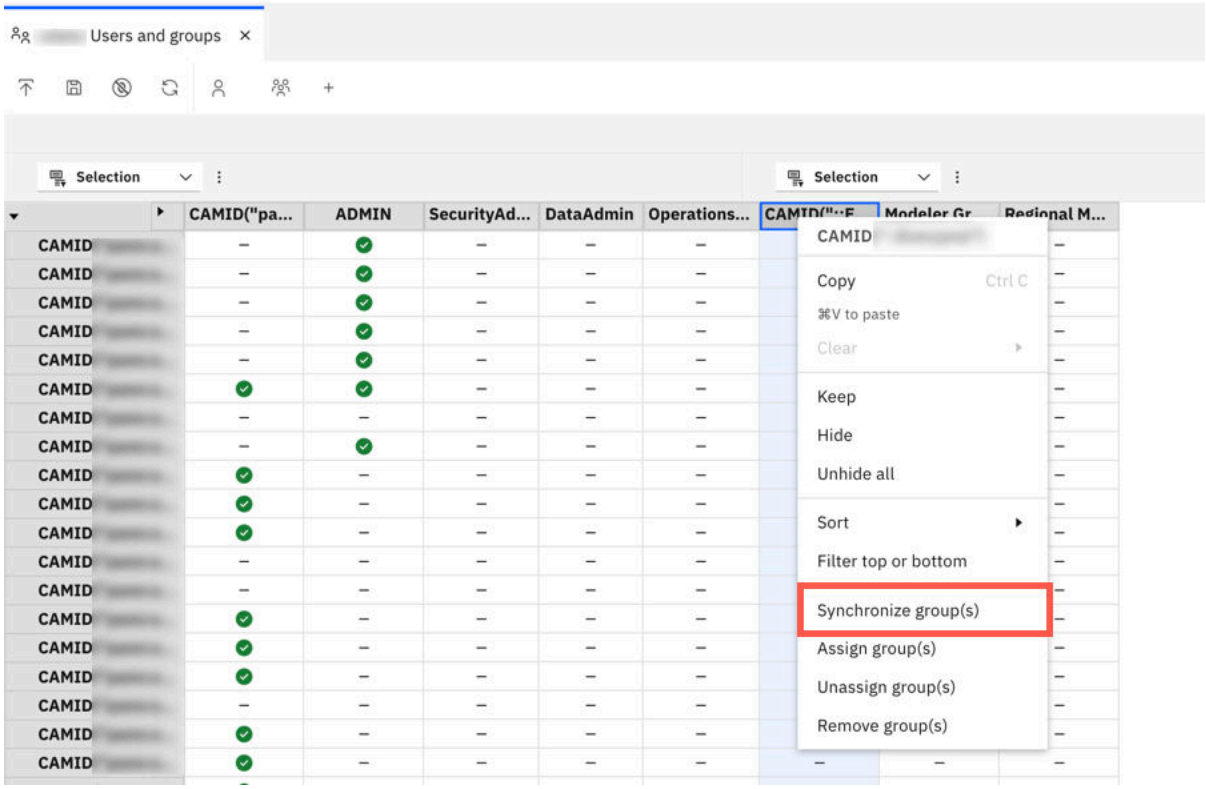
The following new and changed modeling features are available in Planning Analytics Workspace 2.0.89.

## Synchronize user groups in Planning Analytics databases with Planning Analytics Workspace

Administrators and modelers can now synchronize user groups that are in a Planning Analytics database with the groups in Planning Analytics Workspace from the modeling workbench.

**Important:** You cannot synchronize the predefined groups ADMIN, DataAdmin, SecurityAdmin, and OperationsAdmin.

1. In a modeling workbench, right-click a database on the **Databases** tree and select **Manage users and groups**.
2. In the **Users and groups** editor, right-click a group and select **Synchronize group(s)**.




Memberships of the group in the database now match those of the same group that exists in Planning Analytics Workspace. Additionally, when you synchronize a group, existing users in the Planning Analytics Workspace group that are not in the database are also added to the database.

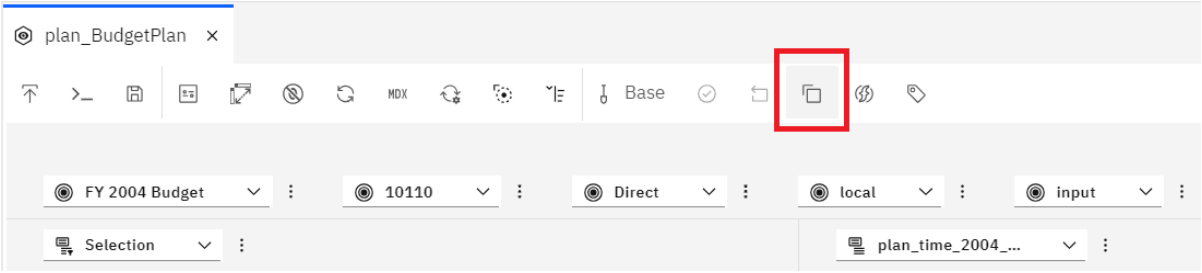
**Note:** Synchronization does not remove any new users that were added to the group in the database.

For more information, see Manage users and groups on a Planning Analytics database (Cloud only).

## Duplicate cubes in the cube viewer

The **Duplicate** button is now available on the cube viewer toolbar on the workbench.

Click on the **Duplicate** button allows to create a duplicate of the current cube view. This button is located between the **Sandbox discard** button and the **Cube rules** button.




The newly duplicated cube view opens in a new tab, and the information in the duplicate includes the latest changes made in the cube.

This feature is also available when working in a pop out window. When duplicating a cube in a pop out window, the duplicated cube opens in a new pop out window.

## Deprecation notices - 2.0.89

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.89.

### Classic cube viewer removed from Planning Analytics Workspace 2.0.90

As of Planning Analytics Workspace 2.0.90, the classic cube viewer is no longer available in Planning Analytics Workspace.

Any existing classic cube viewer will be converted to a new experience cube viewer when first opened in Planning Analytics Workspace 2.0.90.

# 2.0.88 - What's new, June 21, 2023

Learn about new features and known issues in version 2.0.88 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## What's new in books and reporting - 2.0.88

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.88.

### Synchronize widgets at a group level

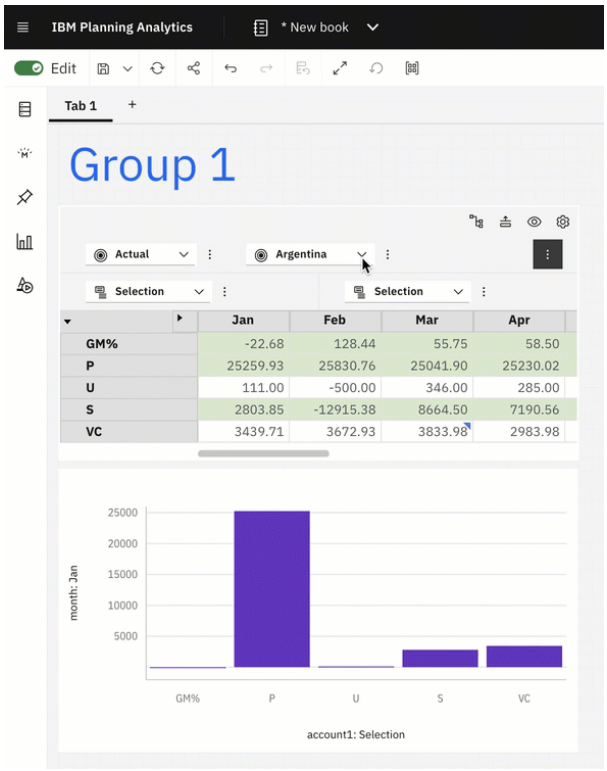
You can now synchronize widgets (explorations, visualizations, synchronization tiles, selector widgets, and websheets) at the group level with group synchronization. Previously, you could set the scope of synchronization at the book or sheet level only.

You can start by creating a synchronization group in book properties.

1. In edit mode, click **Properties**.
2. Under **Synchronization**, click **Create and manage custom groups**.
3. Click **Create** to create a group.
4. Enter a name and description for the group, and click **Create**.

5. Click **Apply changes** to finish creating the group.

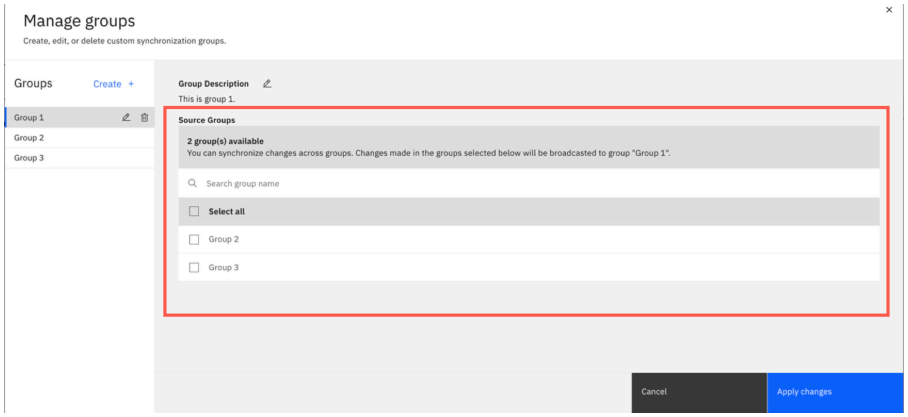
You can now add widgets to your group. Select a widget in your book and under **Properties > Synchronize > Synchronization group**, select the group you just created. Repeat this process for all the widgets that you want to assign to the group. Widgets in the group are now synchronized together.



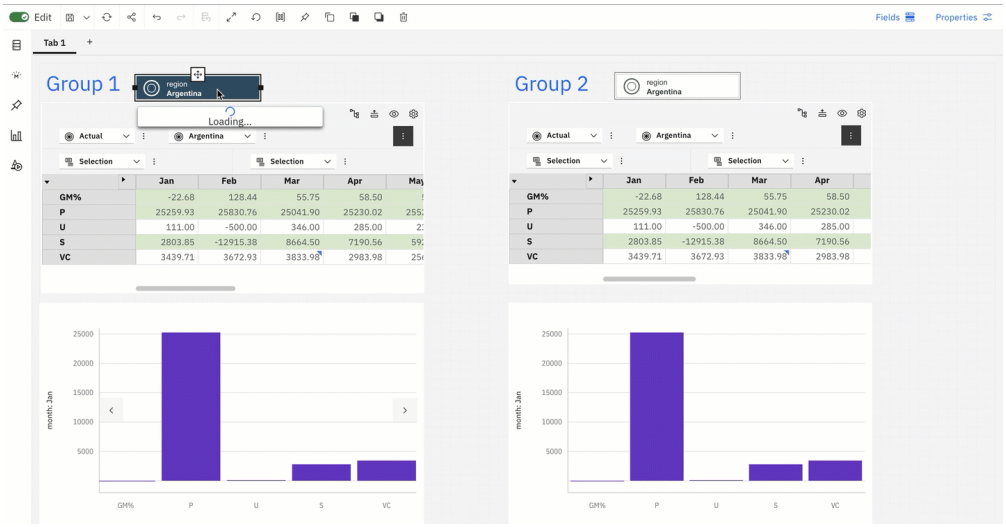

Changes to the currently displayed members of a hierarchy in a widget are shared with other widgets. As members of a hierarchy that are displayed in one widget change, the same members display in synchronized widgets that use the same hierarchy.

Synchronization groups can span to the entire book.

When you have two or more synchronization groups, you can have one group impact another group to achieve an cascading effect. To do this, you can assign more than one source group to a target group in the **Manage groups** dialog.




In the following example, the hierarchy change in Group 1 cascades to all the widgets in Group 1 and Group 2.



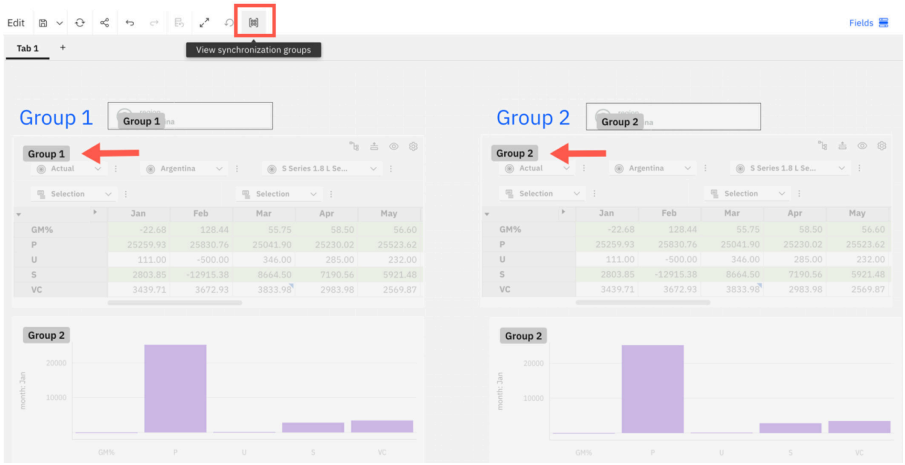

**Note:** Synchronization changes cascade only once. For example, if Group 1 is a source group for Group 2 and Group 2 is a source group for Group 3, changes in Group 1 impact only Group 2.

Additionally, you can now select default synchronization to be at the book, sheet, or group level.

- Book level synchronization - widgets in all the sheets in a book synchronize
- Sheet level synchronization - widgets in the selected sheet synchronize
- Custom group level synchronization - widgets in the book synchronize with changes in the custom group

**Note:** By default, synchronization is set to sheet level.

Click the new **View synchronization groups** icon in the toolbar to view an overlay for all widgets that are part of synchronization. The overlay displays the name of the group the widgets belong to. Click the icon again to interact with the widget.



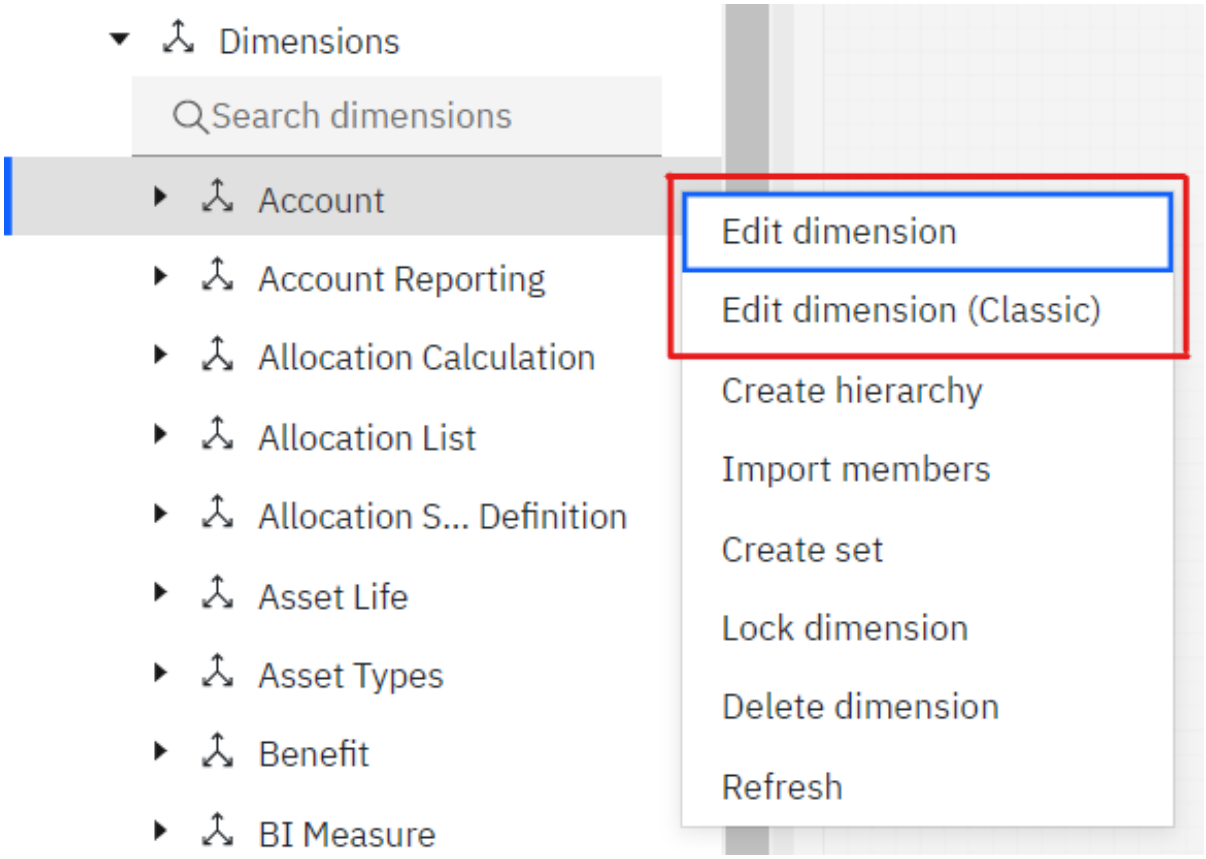

For more information, see Group synchronization.

## Use the new experience dimension editor in books

The new experience dimension editor, which was previously available only on the modelling workbench, is now available in books.

To edit a dimension in the new experience editor, right-click a dimension on the **Databases** tree, then click **Edit dimension**

The classic dimension editor remains available in books. Right-click a dimension, then click **Edit dimension (Classic)** to edit a dimension in the classic editor.



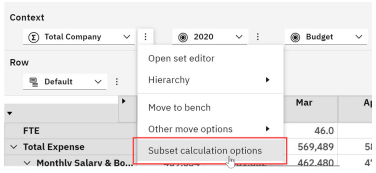

When you save a book with the new editor, you can reopen the book and retain your editor preferences in the same state as in the classic editor.

For more information on the new experience dimension editor, see Welcome to the new experience dimension editor.

## Use subset calculations on the context section of a view

The subset calculation feature in the cube viewer has been added to dimensions on the context section of a view.

To create a subset calculation, click the **Options** menu ⋮ on a context dimension, then click **Subset calculation options**.

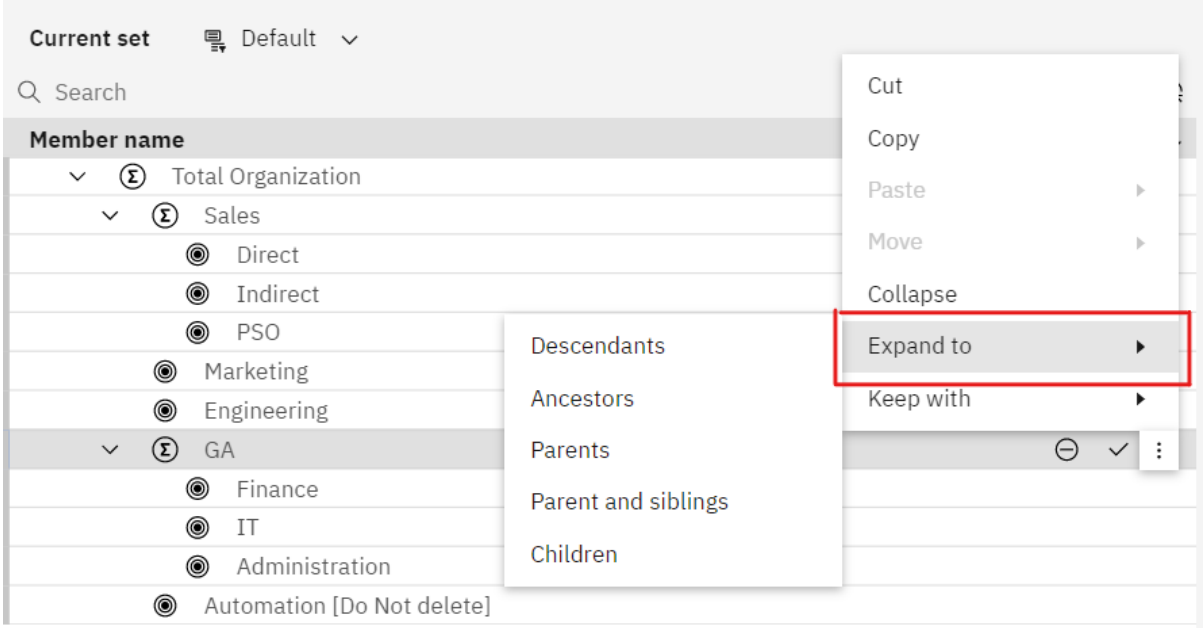



To learn more about subset calculations, see Add a subset calculation to a view.

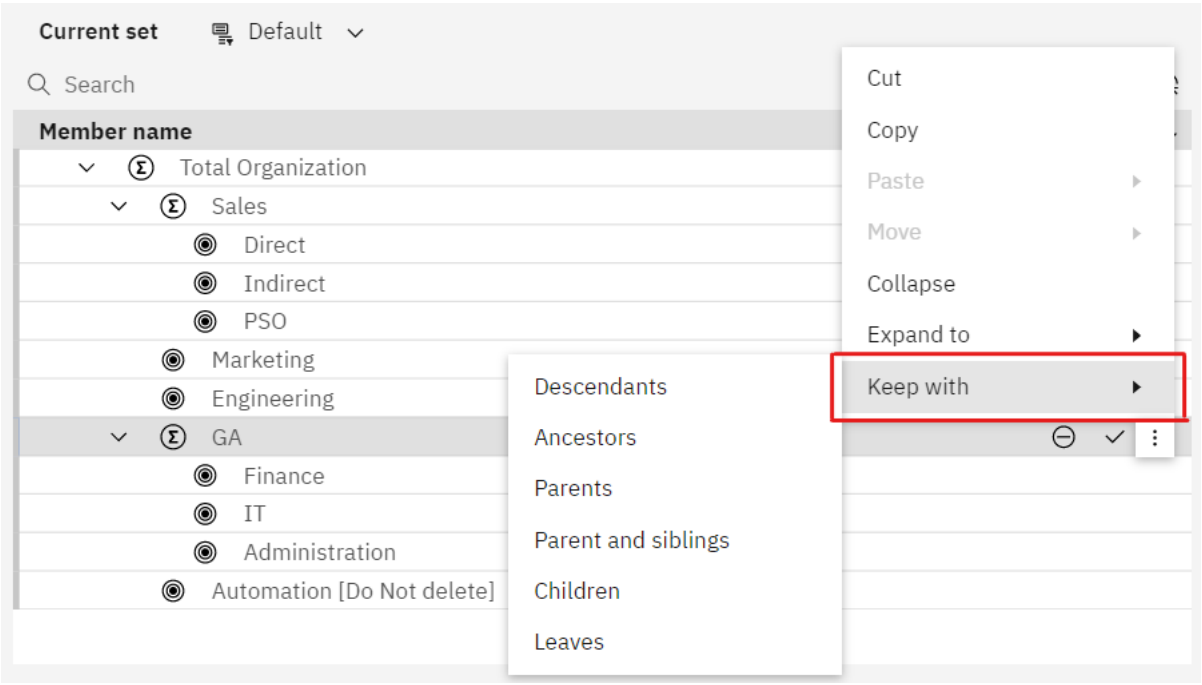
## Use the new 'Expand to' and 'Keep with' options to organize members in a set

**Expand to** and **Keep with** have been added to the set editor. These new features display a set of members, based on hierarchy.

**Expand to** adds members to the set, allowing you to display members of one level higher or one level lower, based on the selected hierarchy. To access this feature, right-click on a member, select **Expand to**, and select the hierarchy you want to display from the menu.




**Keep with**, previously named **Show**, replaces members in the set, based on the selected hierarchy. To access this feature, right-click on a member, select **Keep with**, and select the hierarchy you want to display from the menu.




The **Expand to** and **Keep with** features both have a new menu item, named **Parent and siblings**. In **Expand to**, this allows you to expand a set member to its parents and siblings, and in **Keep with**, this allows you to replace a set member with its parents and siblings.

## What's new in plans and applications - 2.0.88

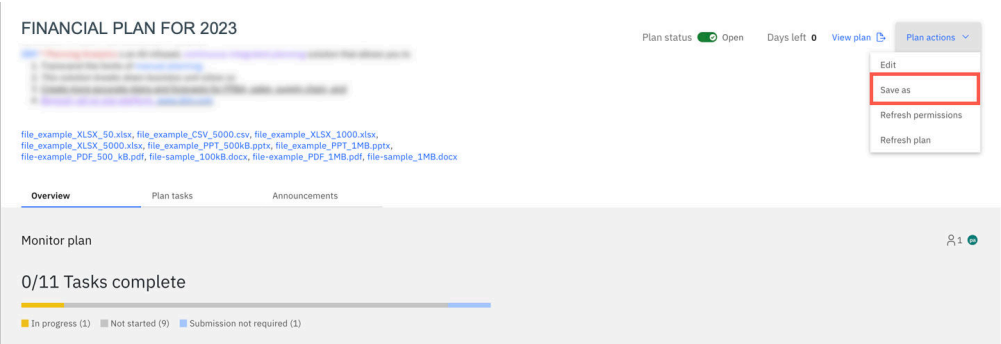
The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.88.

## Duplicate applications and plans with attachments

The duplication feature in Applications and Plans has been enhanced to support attachments. When you duplicate an application or plan, any attachments in the original now automatically copy over to the duplicate. This is also true when you duplicate a section in Applications or task in Plans.

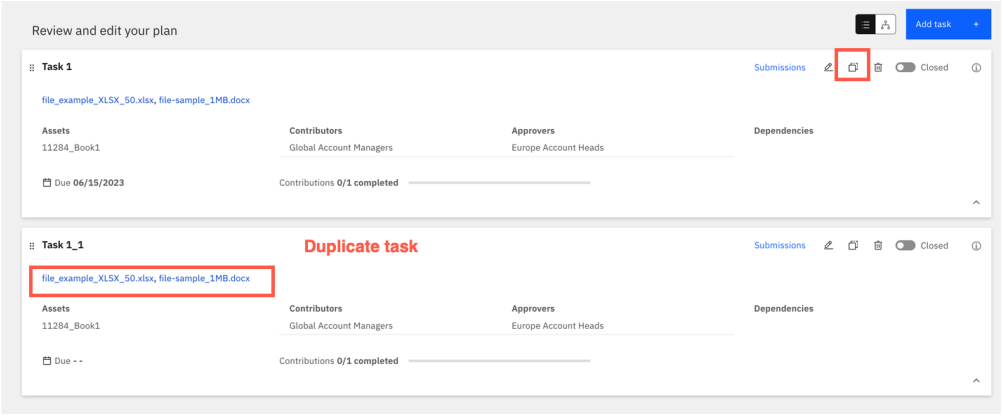
To duplicate an application or plan:

1. From the Home page, click **Applications and Plans**.
2. On the **Applications and Plans** page, click the menu icon next to the plan or application you want to duplicate.
3. Click **Edit**.
4. Under **Application actions** or **Plan actions**, select **Save as**.




5. Rename the duplicate and click **OK**.

To duplicate sections in Applications or tasks in Plans , click the duplicate option in the section or task that you want to duplicate. All attachments in the original task or section now copy over to the duplicate automatically.



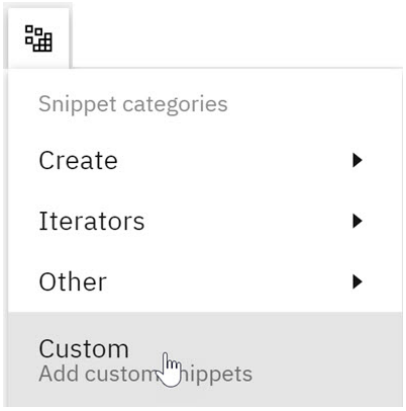

# What's new in modeling - 2.0.88

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.88.

## Create and insert custom snippets for processes

If you have TurboIntegrator process code that you use frequently, you can now save the code as a custom snippet. You can insert a custom snippet into the **Script** tab of a process

When you click the **Snippet** icon on the process editor, a new **Custom** category is available. Select any of the snippets you've created to insert your custom code into a process.




For more information, see Create and upload a custom snippet.

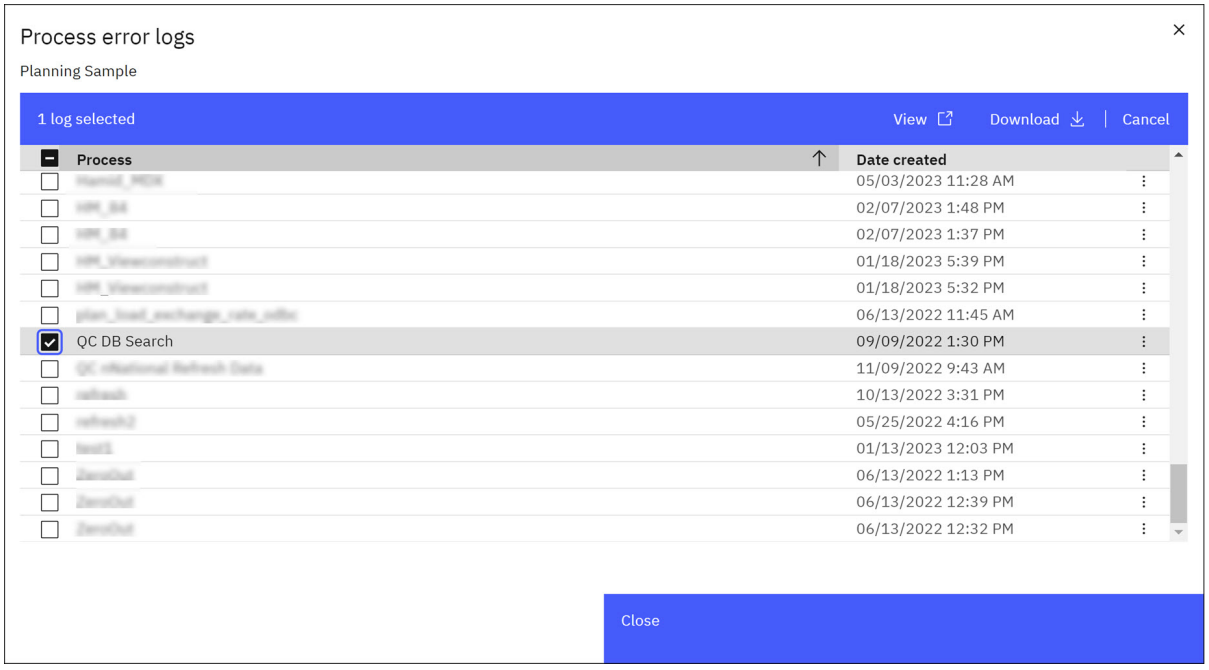
## Access process error logs from the Databases tree

You can view or download process error logs from the **Processes** node on the **Databases** tree in a modeling workbench.

### Procedure

1. On a modeling workbench, expand the database where you want to examine process log files.
2. Right-click the **Processes** node, then click **Error logs**.

   The **Process error logs** page displays all available logs, identified by **Process** name and **Date created**. You can click a column heading to sort the list of logs.




3. Select the individual log that you want to examine.
4. Click **View** to view the log in a temporary viewer or click **Download** to download the error log as a text file.

# What's new in administration - 2.0.88

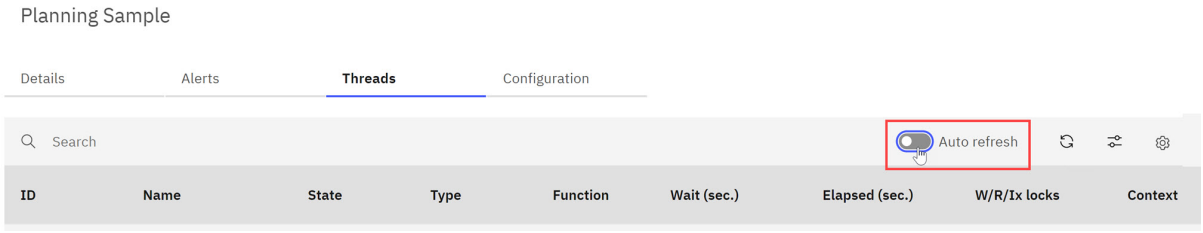
The following new and changed administration features are available in Planning Analytics Workspace 2.0.88.

### Temporarily pause thread or activity refresh

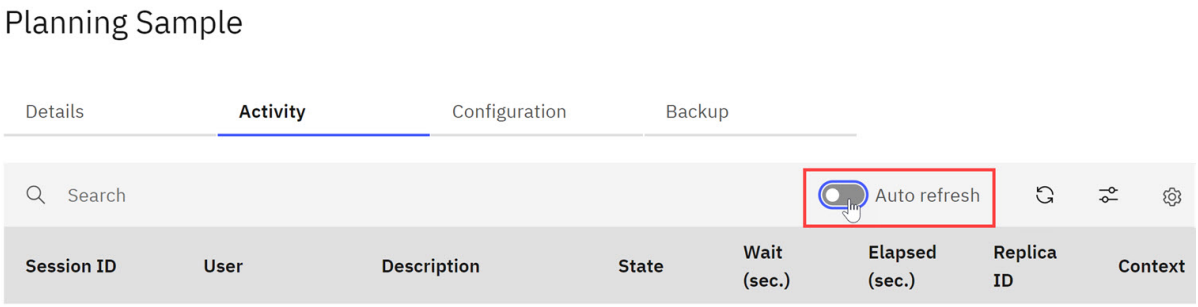
A new **Auto refresh** option on the database page lets you control automatic thread or activity refresh. You can use this option to pause refresh when you want to examine database activity without being interrupted by updates.

The option is available in Planning Analytics Workspace on all platforms, but is located slightly differently according to platform.

- In Planning Analytics Workspace Local and Planning Analytics Workspace on Cloud, the option is available on the database **Threads** tab.




- In Planning Analytics Workspace on Cloud Pak for Data and Planning Analytics Workspace on Planning Analytics as a Service, the option is available on the database **Activity** tab.




To temporarily pause automatic refresh, toggle the **Auto refresh** option to the off position.

Thread or activity collection continues while automatic refresh is paused. You can click **Refresh** to manually refresh the thread or activity list while automatic refresh is paused.

When you toggle the **Auto refresh** option to the off position, the setting does not persist when you leave the **Thread** or **Activity** tab. If you click a different tab or a different database, the **Auto refresh** option is reset to the on position.

## Deprecation notices - 2.0.88

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.88.

### Classic cube viewer removed from Planning Analytics Workspace 2.0.90

As of Planning Analytics Workspace 2.0.90, the classic cube viewer is no longer available in Planning Analytics Workspace.

Any existing classic cube viewer will be converted to a new experience cube viewer when first opened in Planning Analytics Workspace 2.0.90.

## Known issues - 2.0.88

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

### Imported data is not immediately visible in a reserved dimension

In a specific scenario, imported data is not immediately visible in a reserved dimension.

1. Open a dimension on your database.
2. Reserve the dimension.
3. Browse for any import file and import it into the dimension.

The imported data is not visible in the reserved dimension. The members are correctly imported, but not visible while the dimension is reserved.

Release the dimension to view the imported members.

## 2.0.87 - What's new, May 17, 2023

Learn about new features and known issues in version 2.0.87 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

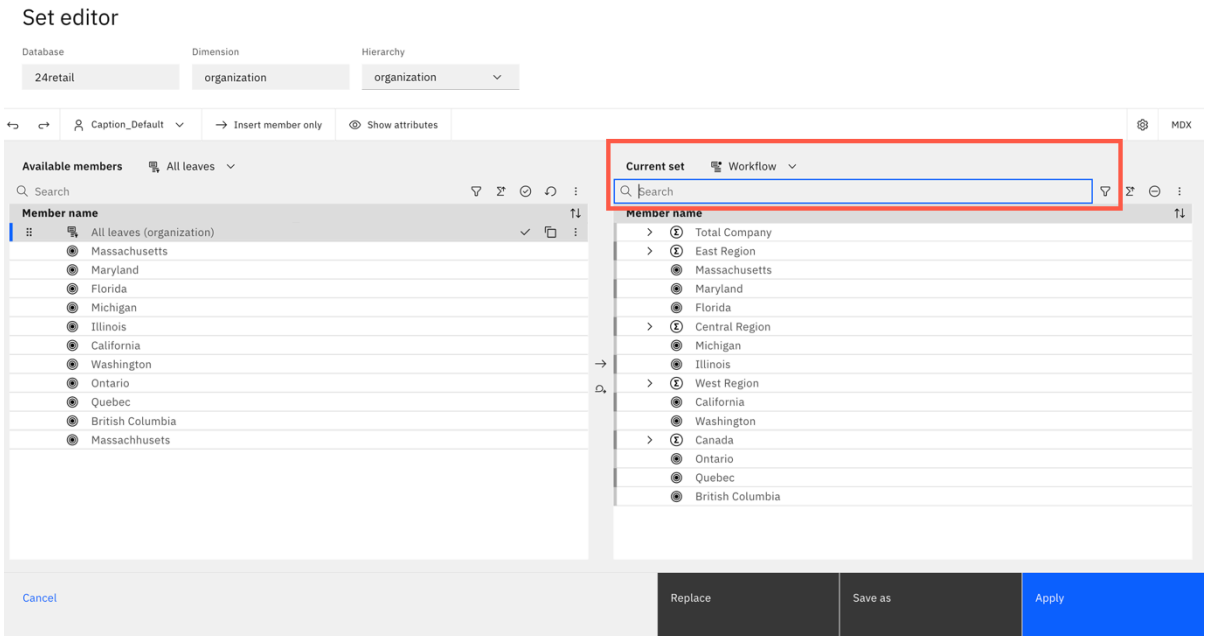
You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

## What's new in books and reporting - 2.0.87

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.87.

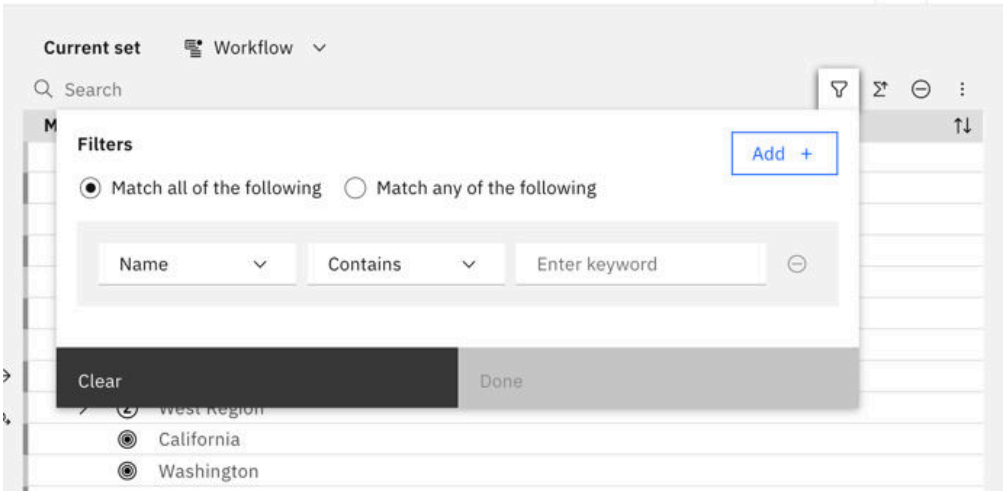
### Search and filter capability available for current set in the set editor

The **Current set** pane in the new experience set editor now has search and filter capabilities. Previously, these features applied only to the **Available members** pane in the set editor.



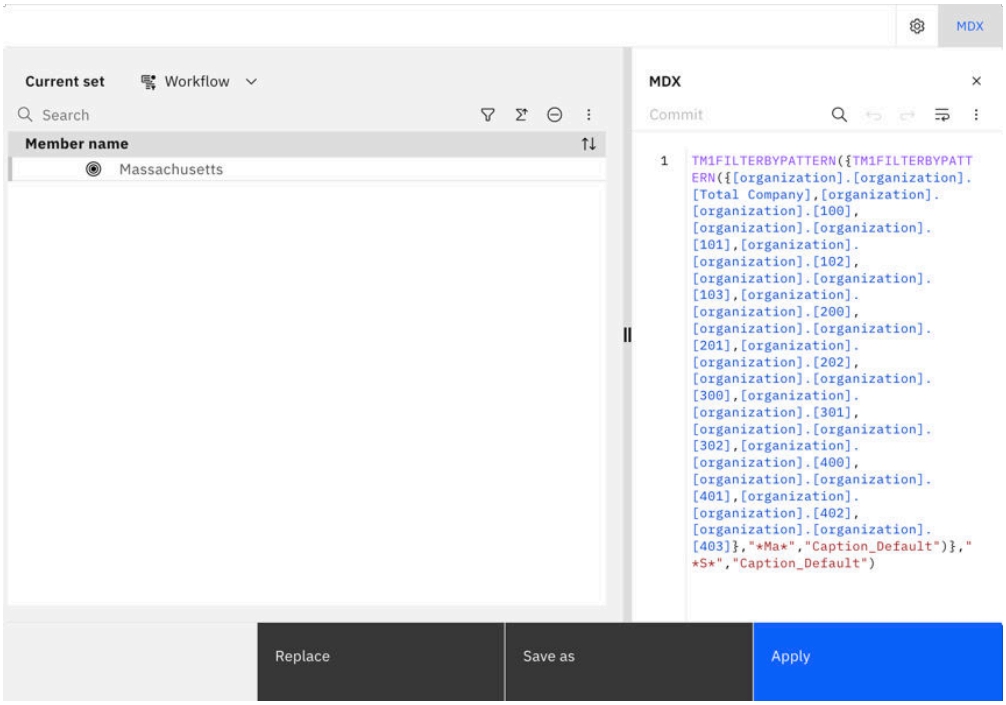

Use **Search** to look for members in the current MDX state for the **Current set**.

To filter the **Current set**, click **Filter** and add your parameters. You can filter a set by up to three parameters at a time.




After you apply a filter, the filter parameter clears from the **Filter** menu, allowing you to add more filters to further narrow your member selections. In an already filtered set, any new filter that you add, applies to the MDX state around the first filter.

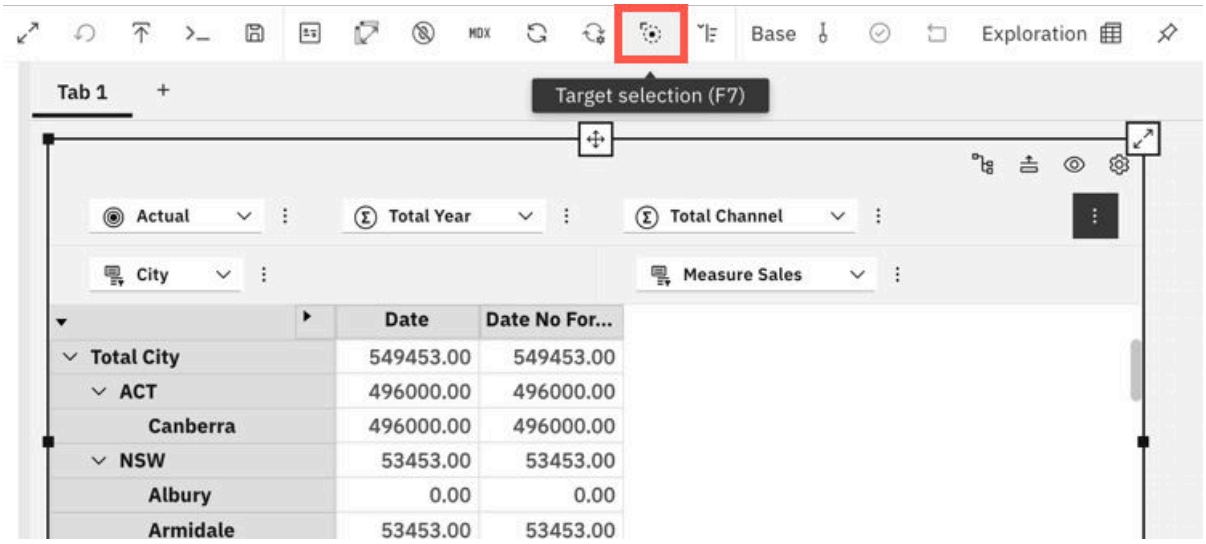
In the following example, the current set was first filtered to only include members with names that contain the letters `ma`. A second filter was applied to narrow down the selections to only include members with names that contain the letter `s`.




At any time, you can click **Undo** to clear your search or applied filters and return to the original set.

## Target selection design and usability improvements

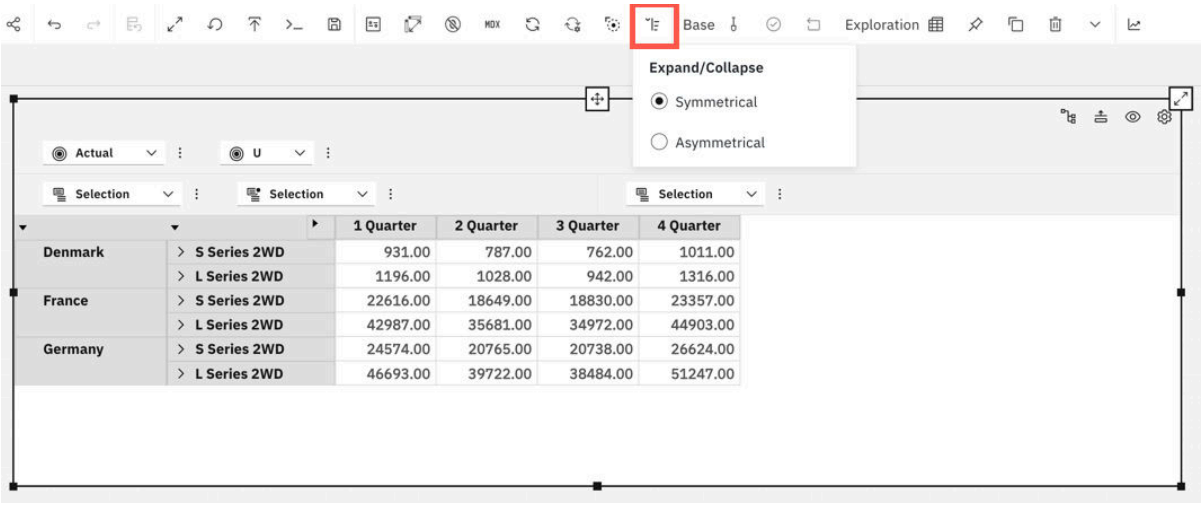
The target selection option is now available on the toolbar. Previously, the **Target selecting** toggle was available on the view grid. You can now also enable target selection mode with the keyboard shortcut **F7**.



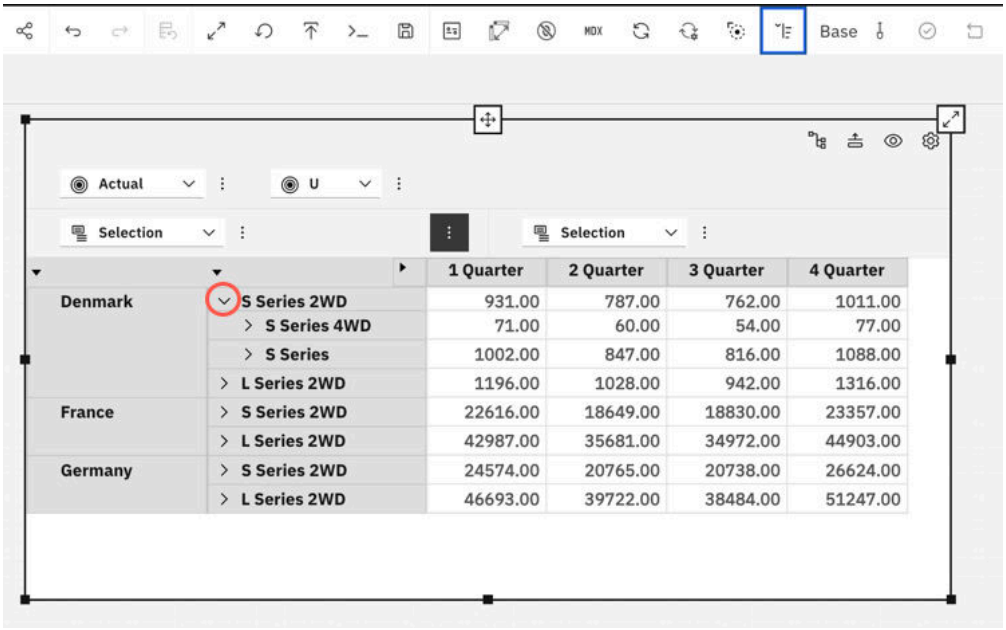

While there is no change to how the target selection feature works, the availability of the target selection option on the toolbar and the new keyboard shortcut makes it easy to use the feature. Simply, click the **Target selection** icon in the toolbar (or press **F7**) to quickly go into target selection mode and make asymmetric selections or calculations.

## Expand and collapse rows and columns asymmetrically or symmetrically

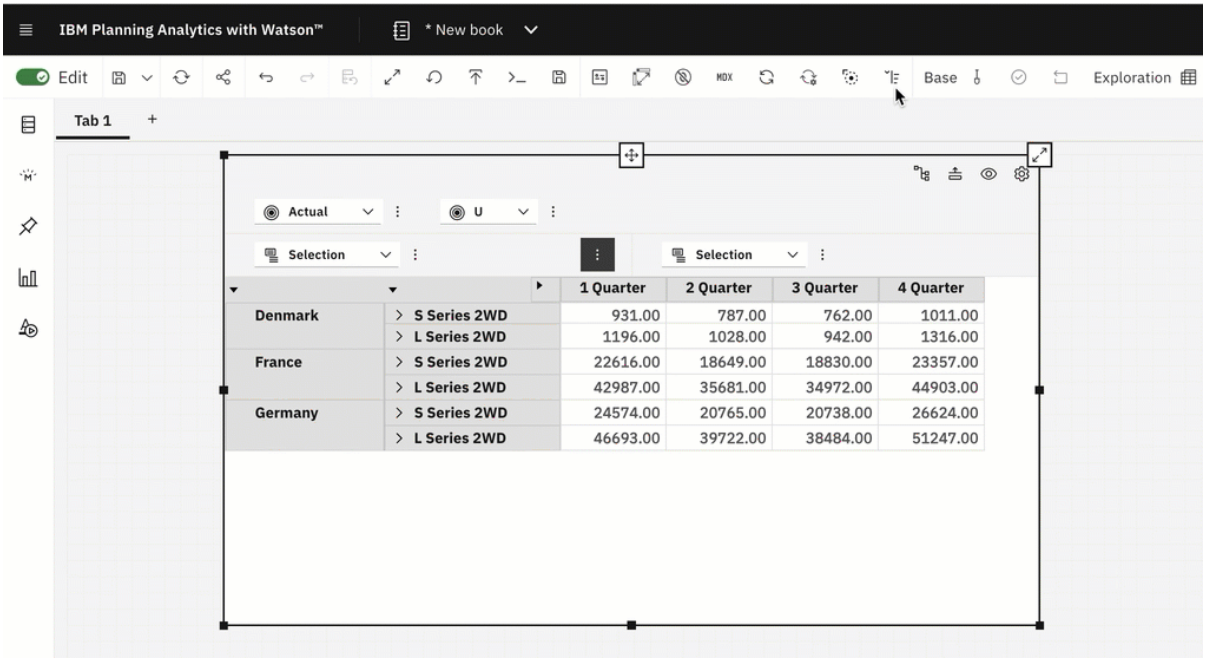
You can now expand and collapse rows and columns in a view either asymmetrically or symmetrically. Click the new **Expand/Collapse** icon in the toolbar and select how you want to expand and collapse rows and columns.




By default, expansion and collapse of rows and columns in views is asymmetrical. This means you can expand or collapse nested members individually.




In symmetrical expansion, expanding a nested member automatically expands that member across all hierarchies. In the following example, notice how expanding `S Series 2WD` for `Denmark` expands `S Series 2WD` for all countries.




**Tip:** If you set **Expand/Collapse** to be symmetrical but need to make an asymmetric gesture, you can do so with **Target selection**. Click **Target selection** or press **F7**, and expand or collapse an individual nested member.

## Zoom bar available in additional visualizations

The zoom bar is now available in several additional visualizations. The zoom bar allows you to scroll and focus on a smaller area of a visualization.

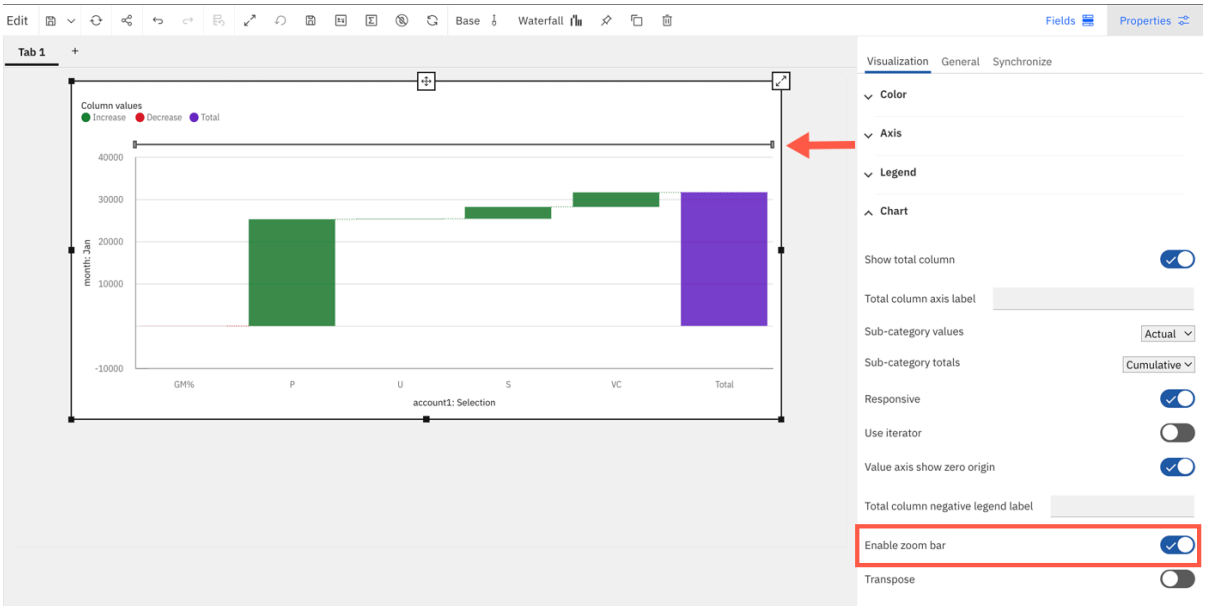
These visualizations now include the **Enable zoom bar** property.

- Box plot
- Marimekko

- Heat map
- Waterfall
- Bullet

The zoom bar was previously available only in the Area, Line, Line and column, Bar, Bubble, Point, Scatter, Stacked bar, Stacked column, and Tree map visualizations.

Use the **Enable zoom bar** property to add the zoom bar to your visualization. Drag a handle on the bar to zoom and focus on an area in the visualization. The area of focus changes as you drag the handle.




**Note:** When you enable the zoom bar, mouse scrolling is disabled in the visualization.

## New properties in the Map visualization

Several new visualization properties were added to the Map visualization in this release.

### World View

The **World View** property displays disputed country borders from the perspective of different countries.

In the **World View** property under **Chart**, select a country and notice how the visualization changes the way disputed country borders display based on the selected country's perspective.

By default, the **World View** property is set to United States (US). Other countries you can choose from are:

- IN - India
- CN - China
- JP - Japan
- AR - Argentina
- MA - Morocco
- RU - Russia
- TR - Turkey

### Icons

You now have the option to select a pre-defined icon for points on a Map. Under **Points layer** in visualization properties, select an icon from the **Icon** property menu. You can also select a custom icon.

**Note:** To use a custom icon, you need to create a custom style and add an icon in Mapbox. You can then import the custom style in Planning Analytics Workspace. For more information, refer to Map.

Use the **Allow overlap** if you want the icons to display on each point even if another element (such as point labels) is conflicting with the icons.

You can hide the default circle icon for points in the Map by setting the **Transparency** property to **100%**.

**Show map labels**

Turn on the **Show map labels** under **Chart** to see background map labels.

## New properties and other changes for the Bullet visualization

The Bullet visualization is updated in this release to include several new properties and changes.

**New categories slot**

A new **Categories** slot under **Fields** lets you draw multiple bullets without the use of **Repeat** slots. The **Categories** slot allows additional sorting capabilities such as sorting by Target and Bar values.

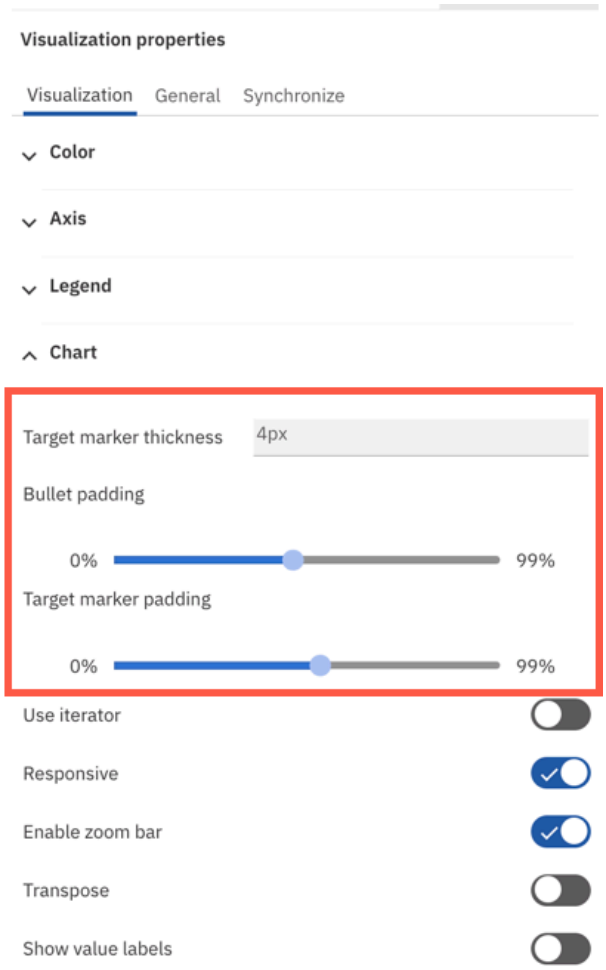
**Style gridlines, axes lines, and axes labels**

Item axis properties and additional value axis properties are now available under **Axis** in **Properties**. You can style gridlines, axes lines, and axes titles and labels. You can also set value axis minimum and maximum values.

**Padding properties**

You can now set bullet and target marker padding under **Chart** in **Properties**.

You can also specify the thickness of the target marker. Target marker thickness is set to 4 px by default.

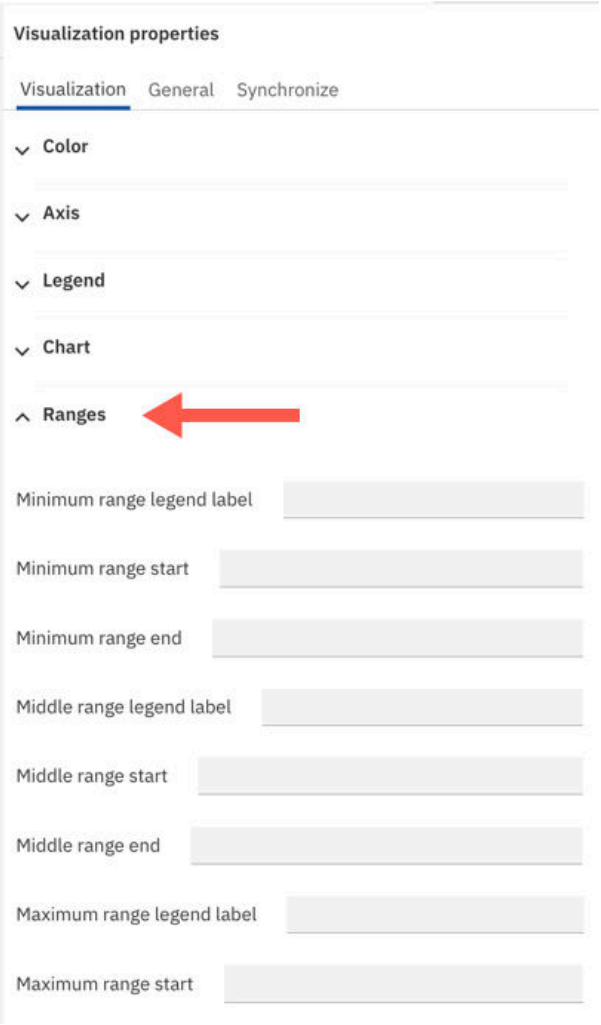



**Zoom bar**

Click **Enable zoom bar** under **Chart** to display a zoom bar in the visualization which allows you to scroll and focus on a smaller area of the visualization.

**Manual range values**

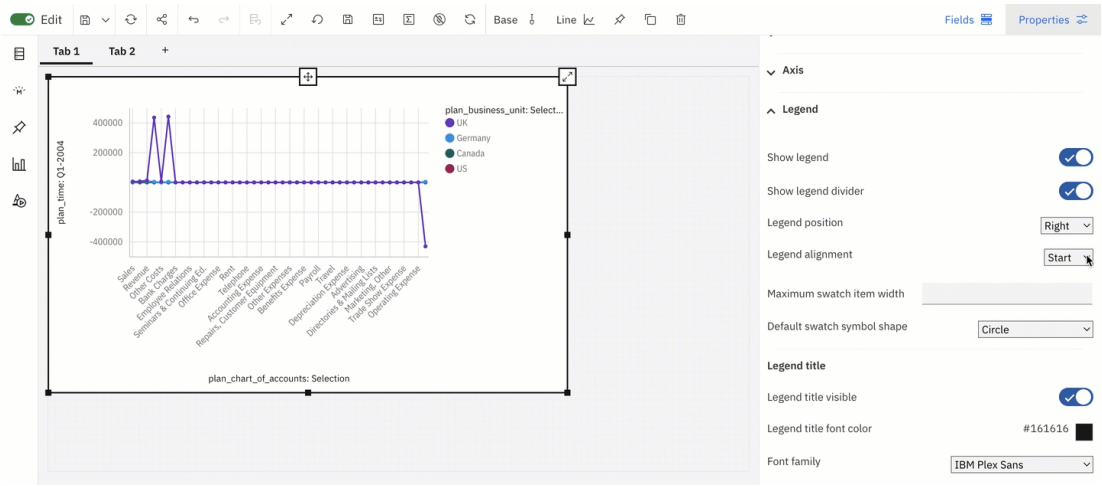
You can now specify minimum, mid, and maximum range values manually under the new **Ranges** section in **Properties**.




## Legend alignment in visualizations

A new **Legend alignment** property in visualizations allows you to position a legend in the desired area of your visualization.

You can now align a legend vertically to display at the start, center, or end of the plot area in visualizations. In combination with the **Legend position**, this new property allows you to position the legend in twelve different areas in the visualization.




To change the alignment of the legend in a visualization:

1. In **Edit** mode, click the visualization.
2. Go to **Properties > Visualization properties > Legend**.
3. Select the desired **Legend alignment**.

## Configuration modification for Learn Pane in Planning Analytics Workspace Local and Cloud Pak for Data

The cognitive Learn Pane in Planning Analytics Workspace Local is now hosted on a new domain. If you are running any version of Planning Analytics Workspace Local prior to 2.0.87, this change requires the following configuration modifications to Planning Analytics Workspace Local and Planning Analytics on Cloud Pak for Data to ensure continued access to documentation and other information through the Learn Pane.

If you upgrade to Planning Analytics Workspace Local 2.0.87 or later, no configuration changes are required to ensure continued access to documentation through the Learn Pane.

**Important:** The change in hosting domain does not interrupt access to content through the Learn Pane for Planning Analytics Workspace on Cloud or Planning Analytics as a Service.

### Planning Analytics Workspace Local Linux and Windows

1. Edit the `<paw_install>/services/glass/docker-compose.yml` to include the environment variable `CCH_URL=https://cchhelp.pac.cloud.ibm.com/docs/cch`.

```
version: '2.1'

services:
  glass:
    image: ${GLASS_IMAGE}
    container_name: glass
    restart: always
    volumes:
    - ${GLASS_LOG_DIR}:/glass-log:z
    environment:
    - CCH_URL=https://cchhelp.pac.cloud.ibm.com/docs/cch
```

   **Important:** The indentation of these new lines must match the indentation of the existing container properties.

2. To apply the change, run `scripts/paw.sh` if you are on Linux or `scripts\paw.ps1` on Windows.

### Planning Analytics Workspace Local Distributed Swarm

1. Edit <paw_install>/paw_2.0.XX/swarm/deployments/paw.yml to find the glass: service definition.
2. Under the glass service properties, add the environment variable CCH_URL=https://cchhelp.pac.cloud.ibm.com/docs/cch.

```
140   glass:
141     image: ${GLASS_IMAGE}
142     networks:
143     - app
144     deploy:
145       mode: replicated
146       replicas: ${REPLICAS_GLASS}
147     logging:
148       driver: "json-file"
149       options:
150         max-size: ${JSON_FILE_MAX_SIZE:-1m}
151         max-file: ${JSON_FILE_MAX_FILE:-10}
152     environment:
153     - CCH_URL=https://cchhelp.pac.cloud.ibm.com/docs/cch
154
```

**Important:** The indentation of these new lines must match the indentation of the existing service properties.

3. Run <paw_install>/start.sh to apply the change.

### Planning Analytics Workspace Local Distributed Kubernetes

1. Edit <paw_install>/paw_2.0.XX/kubernetes/templates/any-node/pa-glass-deployment.yaml to find the glass container definition under "containers:.
2. Add an additional environment variable under the env: section and set the variable CCH_URL=https://cchhelp.pac.cloud.ibm.com/docs/cch as follows:

```
131       containers:
132         - name: glass
133           image: %REGISTRY_PREFIX%/pa-glass:%GLASS_VERSION%
134           ports:
135             - containerPort: 9080
136               protocol: TCP
137           env:
138             - name: CCH_URL
139               value: "https://cchhelp.pac.cloud.ibm.com/docs/cch"
140             - name: ISTIO_ENABLED
141               value: "%PA_KUBE_ISTIO_CONFIGURED%"
```

**Important:** The indentation of these new lines must match the indentation of the existing container environment properties.

3. Run <paw_install>/start.sh to apply the change.

### Planning Analytics on Cloud Pak for Data

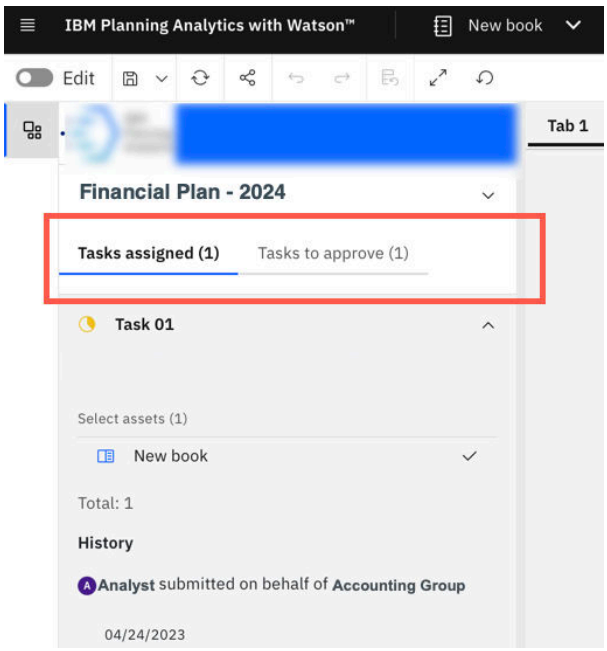
Run the following command: `oc set env deployment glass CCH_URL=https://cchhelp.pac.cloud.ibm.com/docs/cch -n <cpd-instance-ns>`

## What's new in plans and applications - 2.0.87

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.87.

### New tabs on the contribution panel show submission and approval tasks

New tabs on the **Plans contribution panel** display open tasks that are assigned to you for contribution and approval. These new tabs let you quickly submit tasks and approve or reject submissions from one place in the Plan.




You will see these tabs only if you have open tasks that need contribution and approval.

**Note:** After you submit a task or approve or reject a task, the task no longer displays on its respective tab in the contribution panel.

## What's new in modeling - 2.0.87

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.87.

### Use the new "View related cubes" option to identify which cubes include a specific dimension

You can now quickly identify which cubes on a Planning Analyticsdatabase include a specific dimension. This is helpful when you are thinking about deleting a dimension, but want to see the impact of the deletion on the cubes on your database.

#### About this task

The new **View related cubes** option is available only on a modeling workbench. The option is available on the right-click menu anywhere you can see an individual dimension name on the **Databases** tree:

- *DatabaseName* > Dimensions > *DimensionName*
- *DatabaseName* > Cubes > *CubeName* > Dimensions > *DimensionName*
- *DatabaseName* > Control Objects > Control Dimensions > *DimensionName*
- *DatabaseName* > Control Objects > Control Cubes> *CubeName* > Dimensions > *DimensionName*

#### Procedure

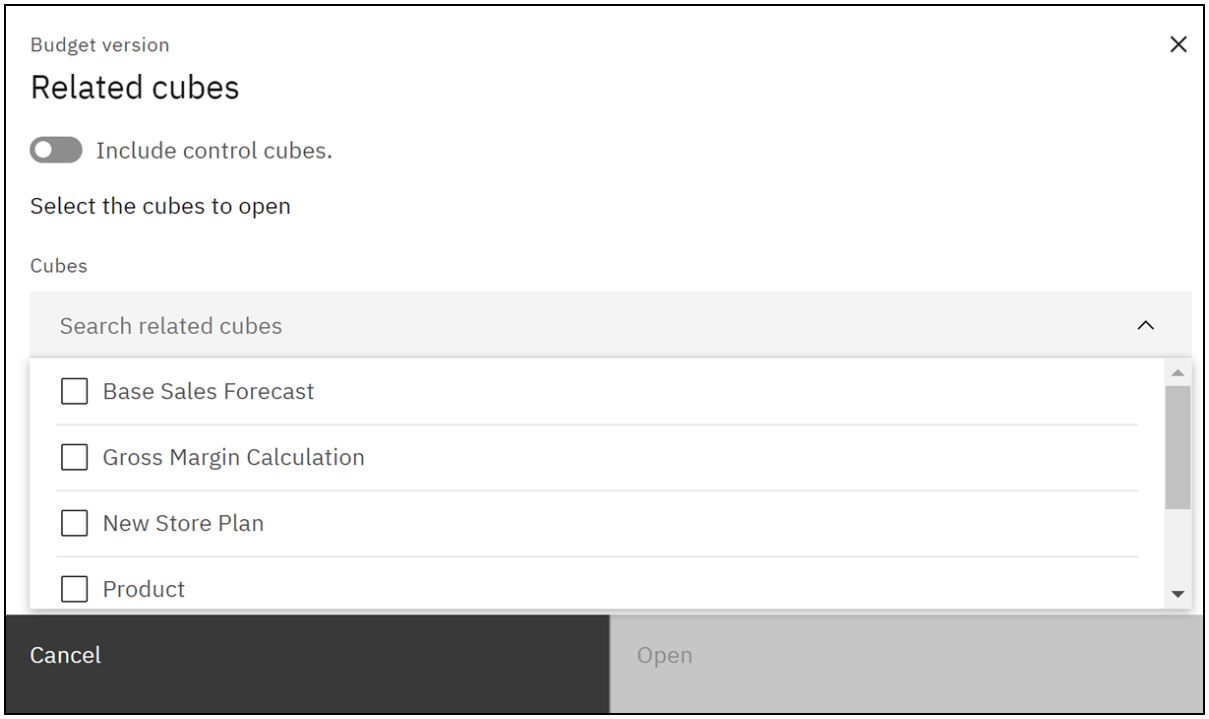
1. Right-click a dimension on the **Databases** tree on a modeling workbench.
2. Click **View related cubes**.

If the dimension is not included in any cubes, you receive notification and can cancel the procedure.

3. On the **Related cubes** window, toggle the **Include control cubes** option to show or hide control cubes in the list of cubes.
4. In the **Cubes** list, select the cubes you want to open for review.

   If the dimension is included in five or fewer cubes, the cubes are shown in a simple list.

   If the dimension is included in more than five cubes, the cubes are shown in a searchable menu list.




5. Click **Open**.

   Each of the cubes you selected opens for review on a separate tab on the modeling workbench.

## Import a data source with multiple data value columns to an existing cube

If a data source includes multiple data value columns, you can now create mappings to import all data columns simultaneously into an existing cube. This is useful when importing data that has multiple value columns for days, weeks, or months.

You cannot import multiple data columns when creating a new cube.

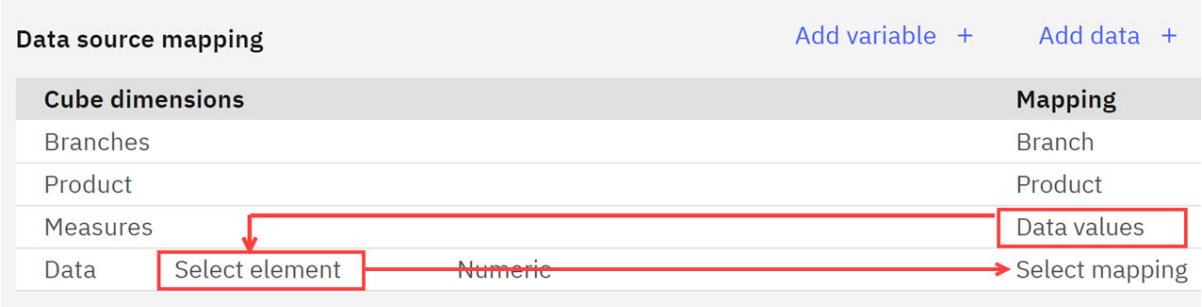
For example, this data source includes value columns for both Quantity and Sales dollars.

Preview

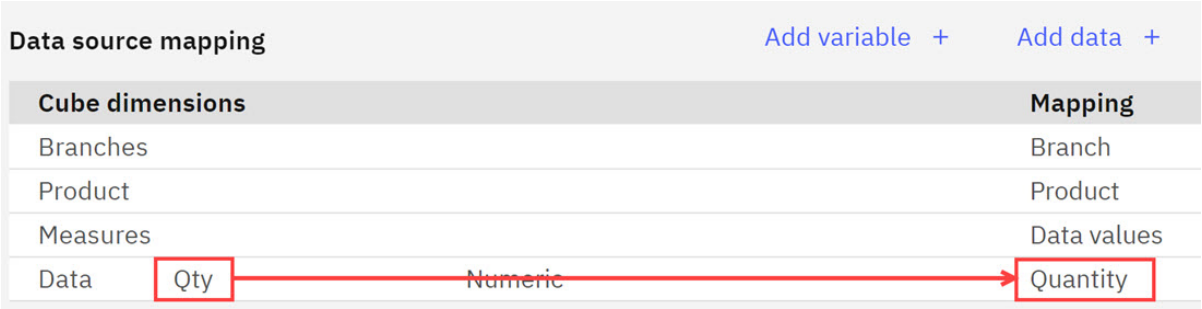
Refresh

| Product | Month | Branch | Quantity | Sales dollars |
|---|---|---|---|---|
| Capri Orange Women's | 15-Nov | Boston | 3976 | 3976 |
| Inferno Blue Men's | 15-Jul | Bury St Edmunds | 2719 | 122355 |
| TX Green Men's | 15-Aug | Leiston | 3078 | 49248 |
| Canyon Mule Climber Backpack Red 32000 cc | 15-Apr | Ipswich | 4801 | 388881 |
| Capri Orange Women's | 15-Sep | Bridgeport | 46 | 46 |
| Glacier Deluxe Black Unspecified | 15-Sep | London | 3839 | 122848 |
| Cat Eye Orange Women's | 15-Jun | Dundee | 3221 | 235133 |
| Pocket Gizmo Red 8.4 cm | 15-Feb | Hamilton | 1215 | 53460 |

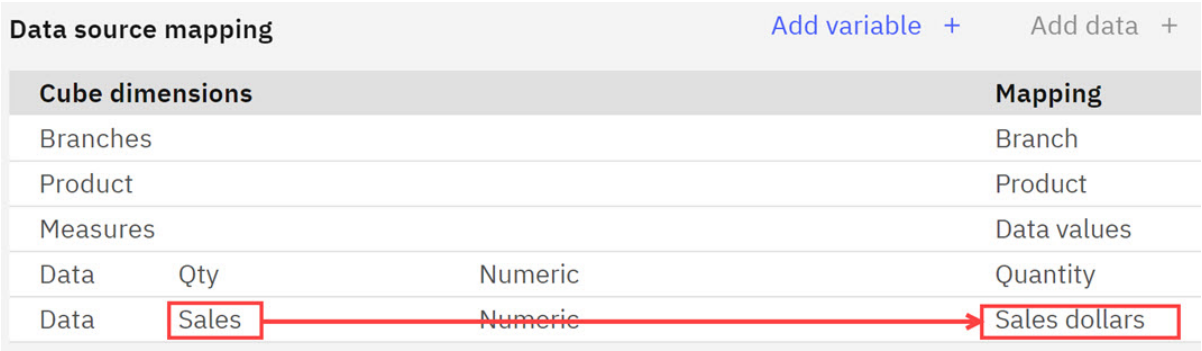
In this case, **Quantity** and **Sales dollars** correspond to elements of the Measures dimension in an existing cube. When you map an existing cube dimension to **Data values** in the data source mapping, a new **Select element** mapping option becomes available, which you can use to map individual elements to value columns in the source.




You can click **Select element** to pick a Measures dimension element in the existing cube, and then click **Select mapping** to map the Measures element to a value column in the data source. In this example, the **Qty** member in the Measures dimension maps to the **Quantity** value column in the data source.




Because the data source has multiple data value columns, you can click **Add data +** to create additional data mappings. Again, using our example, we know there is a **Sales** member in the Measures dimension, you we can map it to the appropriate value column in the data source.




When all the data value columns in the data source are mapped, the **Add data +** option is disabled.

## Use the new editors to manage attributes and security for database objects

New settings editors are available from the **Databases** tree on a modeling workbench, which simplify the task of managing attributes or security for your database objects.

### About this task

Previously, a single settings editor contained multiple interfaces to manage attributes and security for database objects. In Planning Analytics Workspace 2.0.87, a dedicated attributes editor and a dedicated security editor is available for each of these object groups:

- Cubes
- Dimensions
- Processes
- Chores

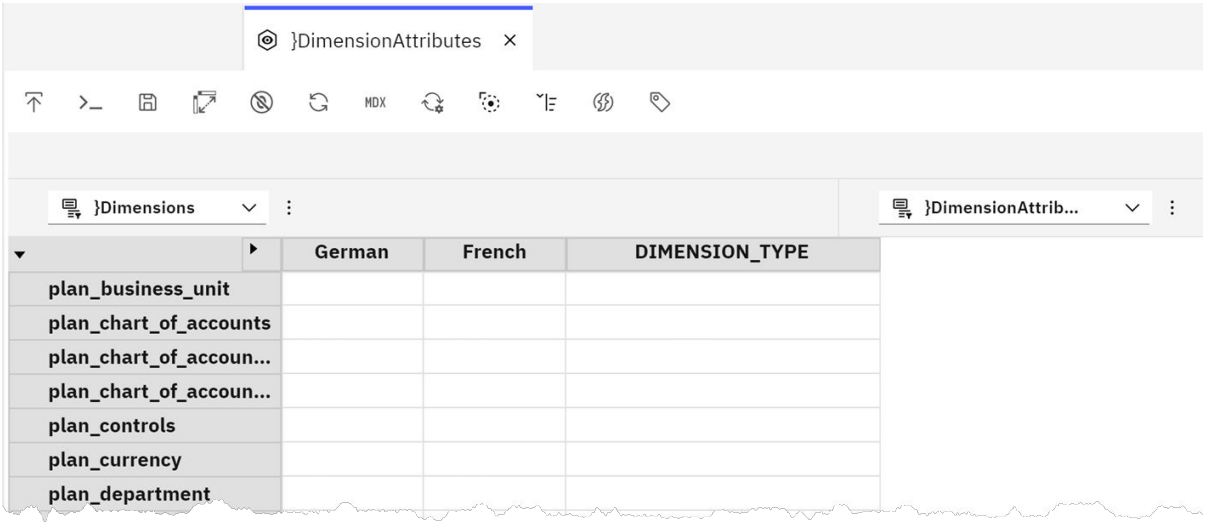
### *Manage attributes for a database object group*

## Procedure

1. On the **Database** tree on a modeling workbench, right-click the object group for which you want to manage attributes: **Cubes**, **Dimensions**, **Processes**, or **Chores**.
2. Click **ObjectName Attributes**. For example, **Cube Attributes** or **Chore Attributes**.

   A modified view of the applicable control cube opens, with individual database objects on the row axis and attributes on the column axis.

   For example, here's the view of the `}DimensionAttributes` cube you see when you click **Dimensions** > **Dimension Attributes**.




3. Click a cell at the intersection of an object and an attribute and enter or modify the attribute value.

   You can use Cut, Copy, and Paste to manage attribute values. You can also use the standard view options to manage sets and to select multiple cells, rows, or columns in the editor.

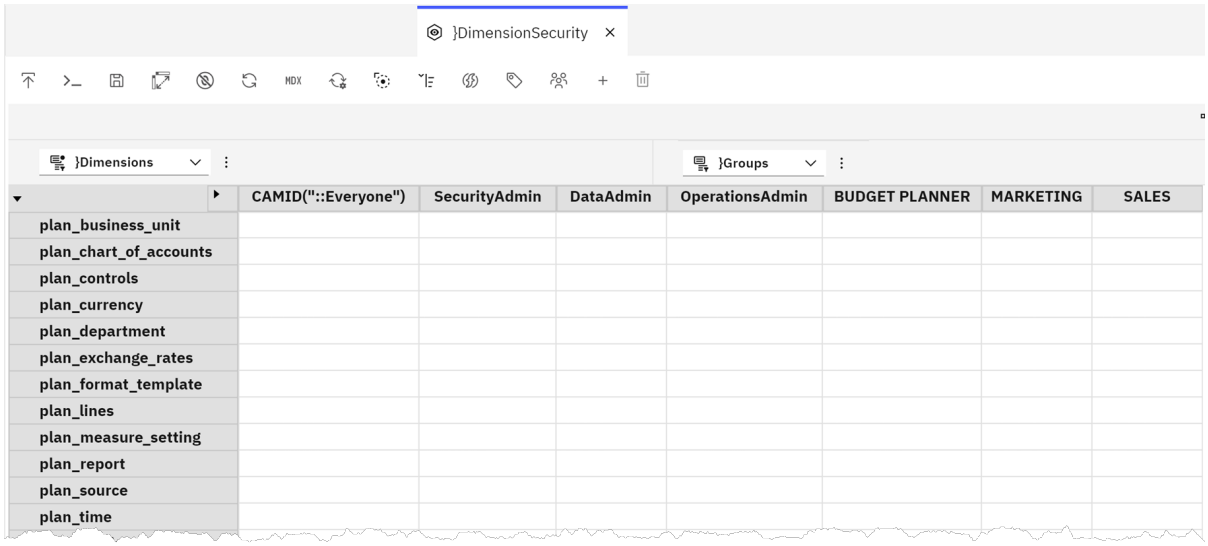
### *Manage security for a database object group*

## Procedure

1. On the **Database** tree on a modeling workbench, right-click the object group for which you want to manage security: **Cubes**, **Dimensions**, **Processes**, or **Chores**.
2. Click **ObjectName Security**. For example, **Cube Security** or **Chore Security**.

   A modified view of the applicable control cube opens, with individual database objects on the row axis and user groups on the column axis.

   For example, here's the view of the `}DimensionSecurity` control cube you see when you click **Dimensions** > **Dimension Security**.




You can use these options to manage security for objects on the Planning Analytics database:

- To assign security for a single user group to a single object, double-click the cell at the intersection of the user group and object, then click the desired security level.
- To assign security to multiple adjacent cells, click the first cell and SHIFT+click the last cell, then right-click the last cell and click **Cell security**.
- To assign security to multiple non-adjacent cells, CTRL+click each cell, then right-click the last cell and click **Cell security**.
- To assign security for an entire row, right-click the element name on the row header, then click **Cell security**. You can also use CTRL+click or SHIFT+click to select multiple rows.
- To assign security for an entire column, right-click the element name on the column header, then click **Cell security**. You can also use CTRL+click or SHIFT+click to select multiple columns.

3. Click + on the security editor to create a new group on the Planning Analytics database. You can then immediately apply object security to the new group.
4. (Cloud only) Click to import groups that exist in Planning Analytics Workspace to the Planning Analytics database.

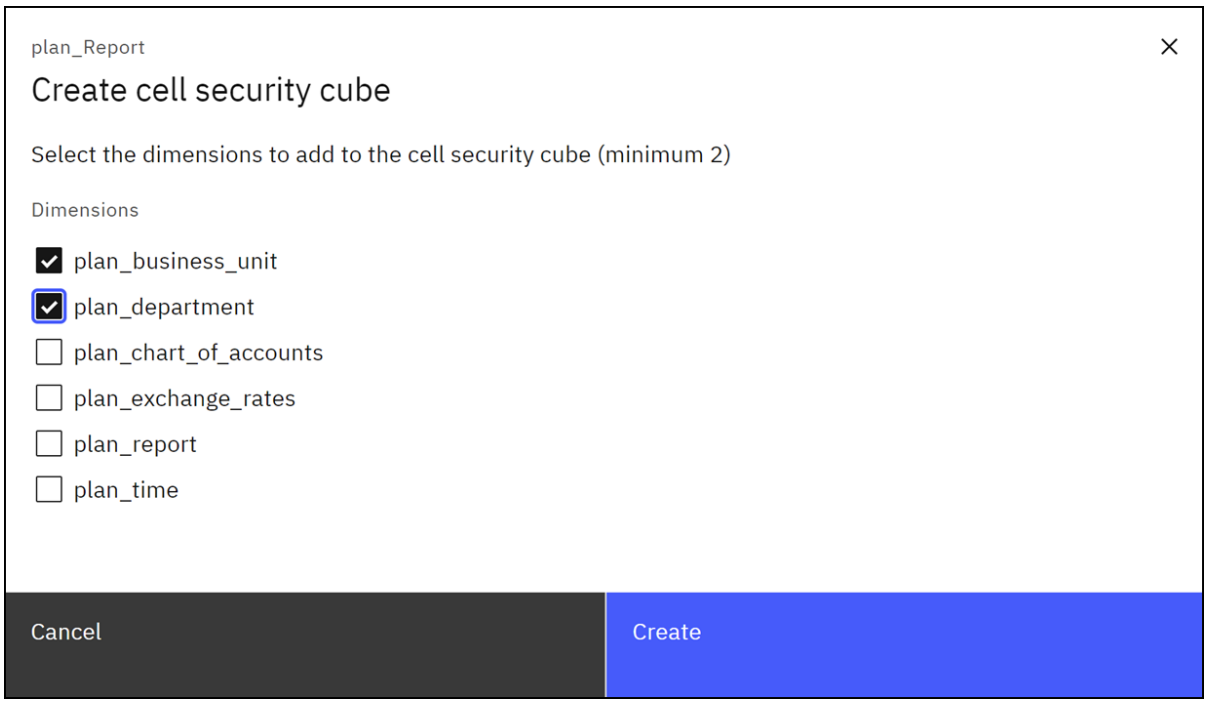
   The groups you select are added to the database. In addition, any users within the groups are also added to the database and assigned to the appropriate groups. You can then immediately apply object security to the new groups.

   You cannot import groups that exist in Planning Analytics Workspace to the Planning Analytics database on Planning Analytics Workspace Local.

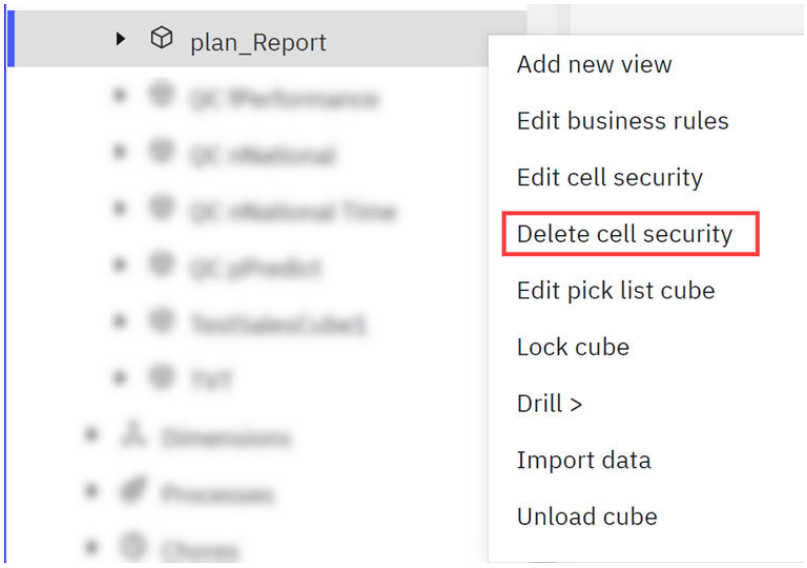
## Manage cell security with a more flexible control cube structure

Previously, a cell security control cube contained all dimension from the data cube, plus the }Groups dimension. In Planning Analytics Workspace 2.0.87, you can select which dimensions to include in a cell security control cube, which can make it easier to manage security.

1. To create cell security, right-click a cube on the **Databases** tree on a modeling workbench, then click **Create cell security**.
2. Select the dimensions you want to include in the cell security cube.
3. Click **Create**.




In addition to the more flexible cell security cube structure, Planning Analytics Workspace 2.0.87 also introduces the ability to delete cell security for a cube.




For more information on defining cell security, see Secure cells.

# What's new in administration - 2.0.87

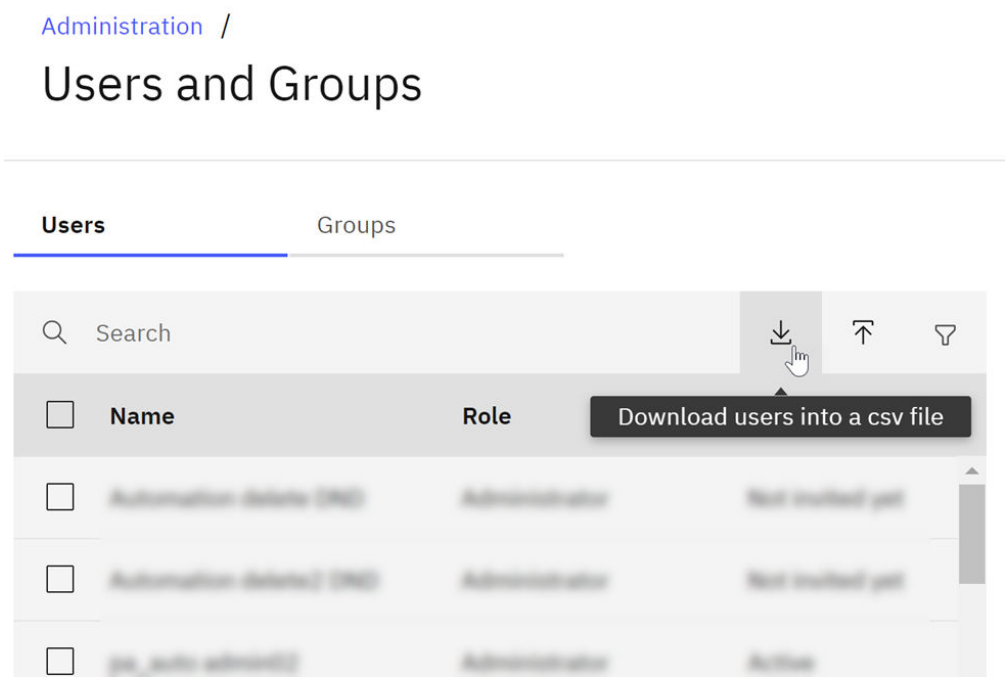
The following new and changed administration features are available in Planning Analytics Workspace 2.0.87.

## View the last login date for Planning Analytics Workspace users

Administrators can now view the last login date for users on their Planning Analytics Workspace environment.

The last login date is not exposed in the Planning Analytics Administration interface, but is included when you download a user report to a .csv file.

1. Open Planning Analytics Administration.
2. Click the **Users and groups** tile.
3. Click **Download users**.




4. Open the downloaded `all-users.csv` file.

The **Last Login** column shows the date and time of the most recent login for the users on your database.

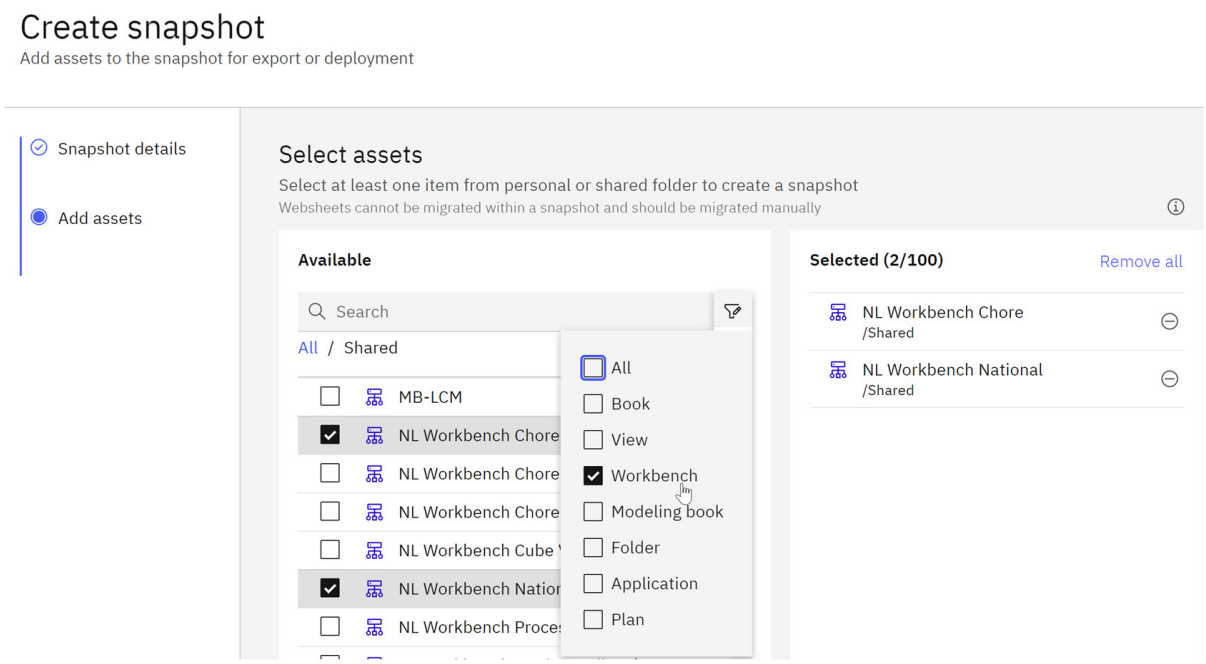
| | A | B | C | D | E | F | G |
|---|---|---|---|---|---|---|---|
| 1 | Login Id | First Name | Last Name | Role | Environment | Directive | Last Login |
| 2 | | | | | | | 2022-06-16T13:15:41Z |
| 3 | | | | | | | 2022-06-22T22:36:26Z |
| 4 | | | | | | | 2022-06-23T13:44:17Z |
| 5 | | | | | | | 2022-06-23T15:33:16Z |
| 6 | | | | | | | 2022-06-23T18:35:09Z |
| 7 | | | | | | | 2022-06-23T19:06:10Z |
| 8 | | | | | | | 2022-06-24T13:23:02Z |
| 9 | | | | | | | 2022-09-21T16:02:35Z |
| 10 | | | | | | | 2022-09-22T12:33:14Z |
| 11 | | | | | | | 2023-01-03T19:36:17Z |
| 12 | | | | | | | 2023-02-06T18:36:23Z |
| 13 | | | | | | | 2023-02-06T18:36:29Z |
| 14 | | | | | | | 2023-02-09T16:42:18Z |
| 15 | | | | | | | 2023-02-17T20:25:18Z |
| 16 | | | | | | | 2023-04-21T12:15:59Z |
| 17 | | | | | | | 2023-04-25T18:27:53Z |
| 18 | | | | | | | 2023-04-26T15:42:16Z |
| 19 | | | | | | | 2023-04-26T15:42:16Z |
| 20 | | | | | | | 2023-04-26T15:45:05Z |
| 21 | | | | | | | 2023-04-26T15:45:05Z |
| 22 | | | | | | | 2023-04-27T17:32:59Z |
| 23 | | | | | | | 2023-04-27T17:33:33Z |

The **Last Login** column is ignored if you use the `all-users.csv` file to upload users to a database.

## Include modeling workbenches in Lifecycle Management snapshots

You can now include modeling workbenches when you use Lifecycle Management to copy and deploy Planning Analytics assets.

When you create a snapshot, the **Workbench** asset type is available when you filter or choose assets for inclusion in the snapshot.




For more information on using Lifecycle Management, see Copy and deploy assets with Lifecycle Management in Planning Analytics Workspace.

# 2.0.86 - What's new, April 18, 2023

Learn about new features and known issues in version 2.0.86 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

## What's new in books and reporting - 2.0.86

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.86.

### Synchronize same hierarchies across different databases

You can now synchronize same hierarchies that are in different Planning Analytics databases.

For example, when you enable synchronization for the same hierarchy that is in two different databases, changes you make to the data in one widget is reflected in the synchronized widget.

**Important:** Sandbox hierarchies do not synchronize across different databases.

To enable same hierarchy synchronization across different databases:

1. In edit mode, create a book with two views from cubes that are in two different databases.
2. Select a view and go to **Properties > Synchronize**.
3. Under **Hierarchies**, enable **Synchronize hierarchies** to synchronize all hierarchies.

All hierarchies are selected by default. You can turn off synchronization for a hierarchy if you want to exclude it from the synchronization.

4. Repeat steps 2 and 3 for the second view.
5. Click **Properties** again to close the panel.
6. Click in the book and go to **Properties > Synchronize**.
7. Under **Mappings**, turn on **Synchronization across servers**.

Make a change in one of the views to see the data update in the other view.

## Disable select data spreading methods on Planning Analytics Workspace Local

You can configure your Planning Analytics Workspace Local environment to disable any combination of the **Repeat**, **Repeat leaves**, **Equal leaves**, and **Straight line** data spreading methods.

### About this task

When you disable a data spreading method, the method cannot be applied in a cube view or in a single cell widget. The method is grayed-out and cannot be selected in the **Data spread options** window and the spreading method cannot be applied using data spread keyboard shortcuts.

**Important:** This procedure disables data spreading only in a cube view or in a single cell widget in Planning Analytics Workspace. If Planning Analytics for Microsoft Excel is also available in your local environment, these settings have no impact on data spreading in Planning Analytics for Microsoft Excel.

### Procedure

1. Open the Planning Analytics Workspace Local configuration file.

   The configuration file is located in the `<planning_analytics_install_dir>\config` directory and is named `paw.ps1` (Windows) or `paw.env` (Linux).
2. Add the **`DISABLED_SPREAD_OPTIONS`** parameter to the configuration file and specify the data spreading methods you want to disable.

   The methods you can specify for this parameter are:

   - Repeat
   - RepeatLeaves
   - EqualLeaves
   - StraightLine

   If you specify multiple data spreading methods, separate each method with a comma and enclose the entire parameter value string in quotation marks as shown in the example.
3. Save the configuration file.
4. Run `./scripts/paw.ps1` (Windows) or `./scripts/paw.env` to apply the configuration changes.

#### Example

To disable the Repeat, Repeat leaves, Equal leaves, and Straight line data spreading methods on Windows, add this line to the `paw.ps1` configuration file.

```
$env:DISABLED_SPREAD_OPTIONS="Repeat,RepeatLeaves,EqualLeaves,StraightLine"
```

To disable the Repeat, Repeat leaves, Equal leaves, and Straight line data spreading methods on Linux, add this line to the `paw.env` configuration file.

```
export DISABLED_SPREAD_OPTIONS="Repeat,RepeatLeaves,EqualLeaves,StraightLine"
```

# What's new in plans and applications - 2.0.86

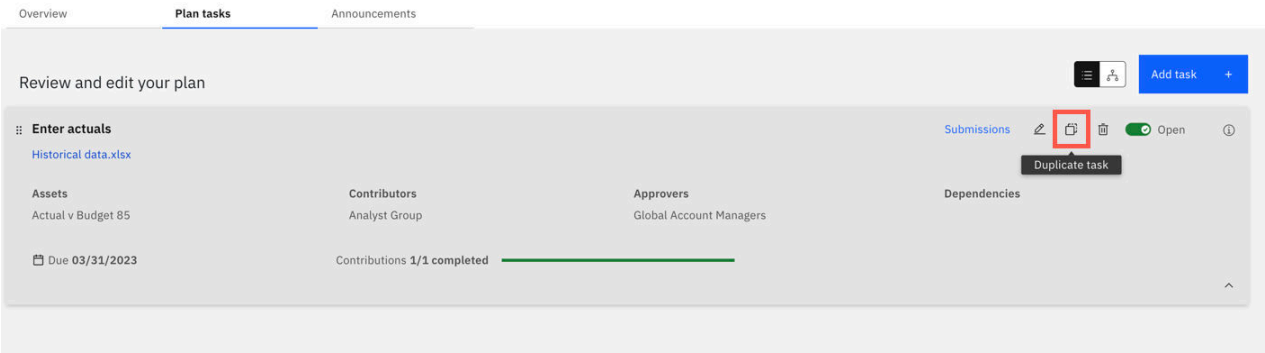
The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.86.

## Duplicate tasks and sections in Plans and Applications

You can now duplicate tasks in Plans and sections in Applications to quickly create similar tasks and sections, saving you time.

### Duplicating tasks in Plans

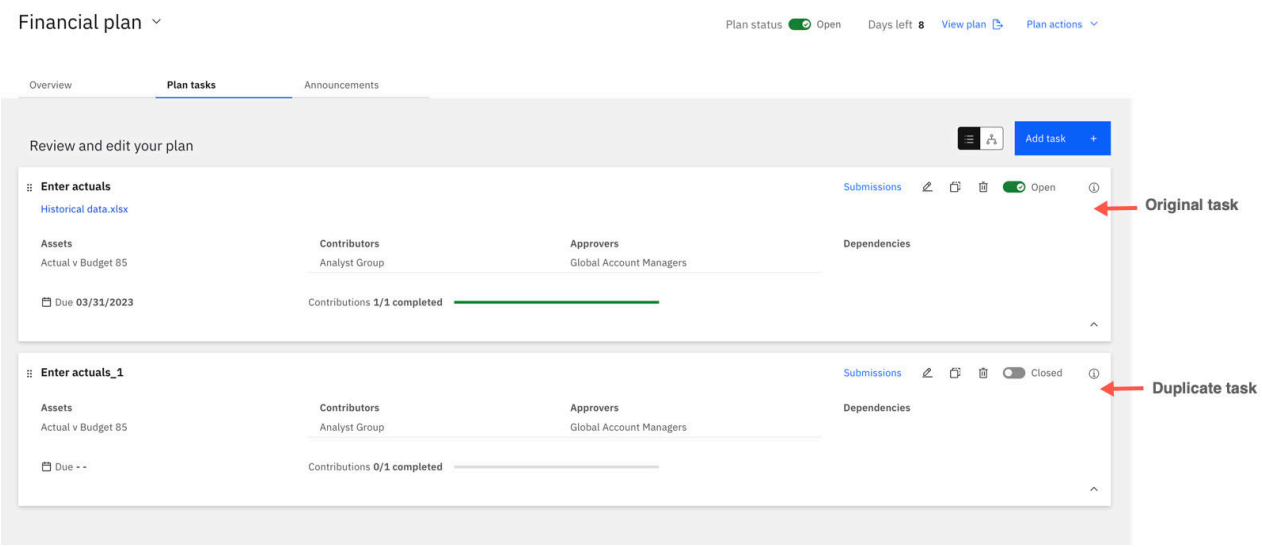
In list view on the **Plan tasks** tab, click **Duplicate task** for the task you want to duplicate.




You can also duplicate tasks in the flow view by selecting the task and clicking **Duplicate**.

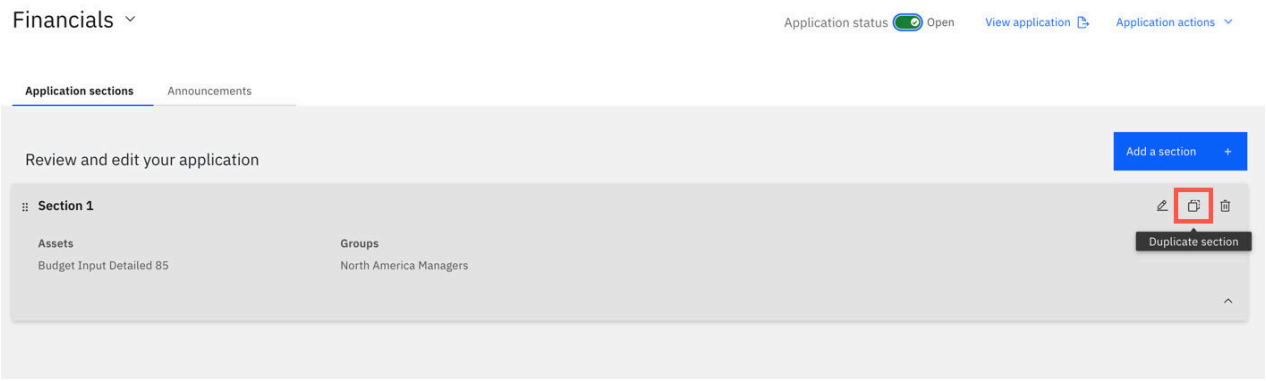
When you duplicate a task, most of the details such as the dependencies, contributors, and approvers are copied from the original task to the duplicated task. The following information in the duplicate task differs from the original task:

- Task name - The duplicated task name includes the original name and appends an underscore and number at the end of the task name.
- Start and due dates - Dates that occur in the past are reset. For the start and due dates to be copied over to the duplicated task, the start date must occur in the future.
- Attachments - Attachments are not copied to the duplicated task.
- Submission logs - Contributions and approvals are not copied over and submission logs are reset in the duplicated task.
- Task status - The task status for duplicated tasks is set to **Closed**.



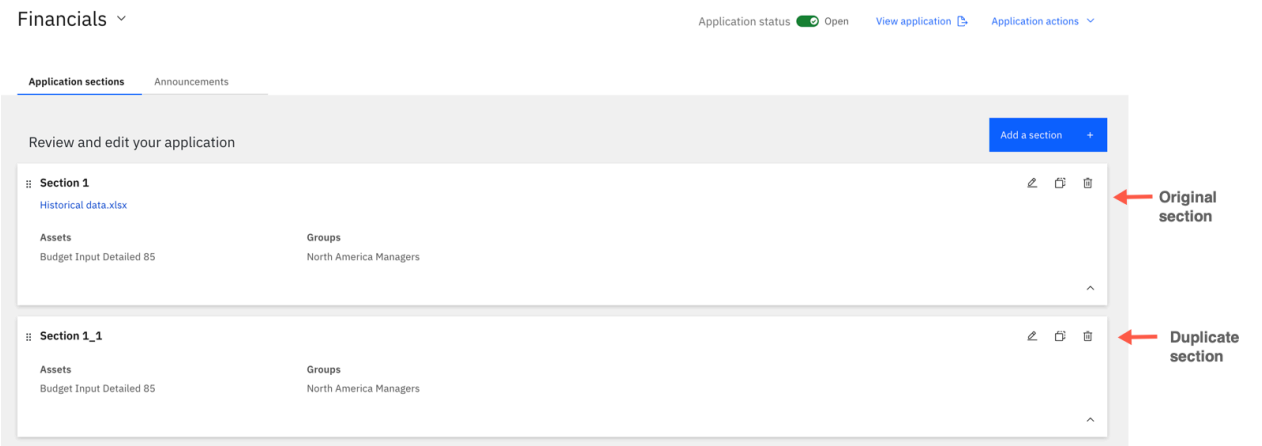

## Duplicating sections in Applications

Under **Application sections**, click **Duplicate section** to duplicate a section that you have already created in your application.



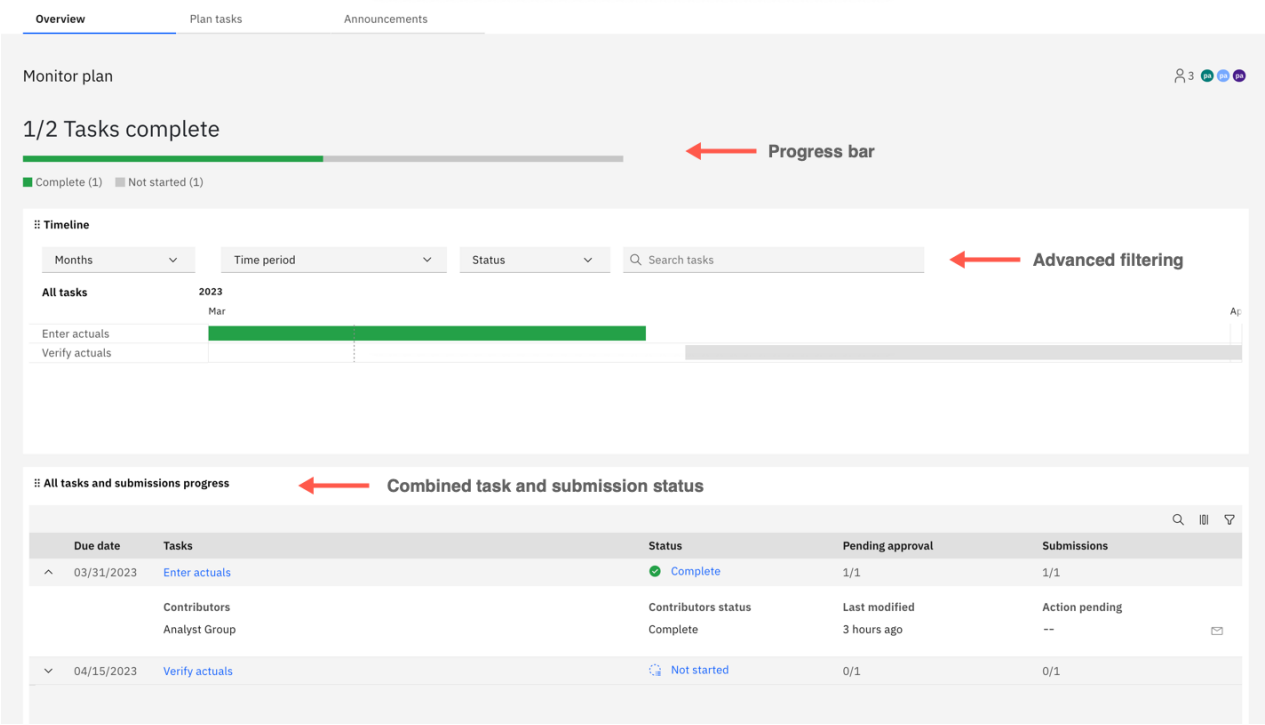

When you duplicate a section, details such as the **Assets** and **Contributor** groups are copied over to the duplicated item. The following information in the duplicate section is different from the original:

- Section name - The duplicated section name includes the original name and appends an underscore and number at the end of the section name.
- Attachments - Attachments are not copied from the original section to the duplicated section.



## New look and usability improvements for Plans Overview

In Planning Analytics Workspace 2.0.86, the **Overview** tab in Plans features a new design and several usability improvements that help you quickly see your plan's progress.




## New progress bar

A progress bar now features on the **Overview** tab that displays the progress of your plan.

## Advanced filtering in the Timeline panel

The **Timeline** panel now includes advanced filtering. You can filter tasks by name, task status, and time period.

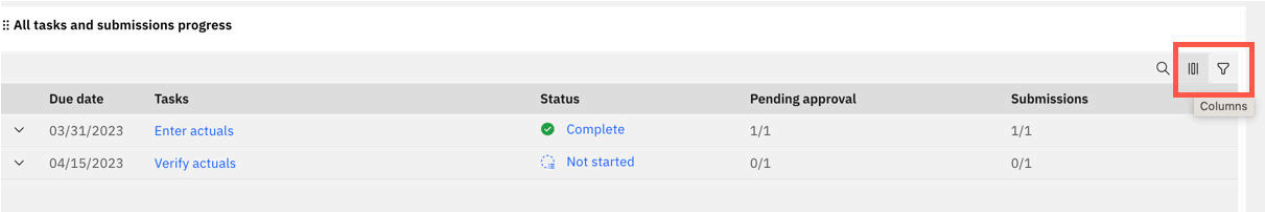
## New All tasks and submissions progress panel

Previously, plan details were split into three panels on the **Overview** tab: **Plan progress**, **Contributions**, and **Timeline**.

The **Plan progress** and **Contributions** panels are now combined into the **All tasks and submissions progress** panel, which allows you to see the progress of your plans' tasks and its submissions in one place.

Expand each task to see status details for the submissions in that task.

You can customize the **All tasks and submissions progress** panel by choosing the columns you want to see and filtering by task status.



# What's new in modeling - 2.0.86

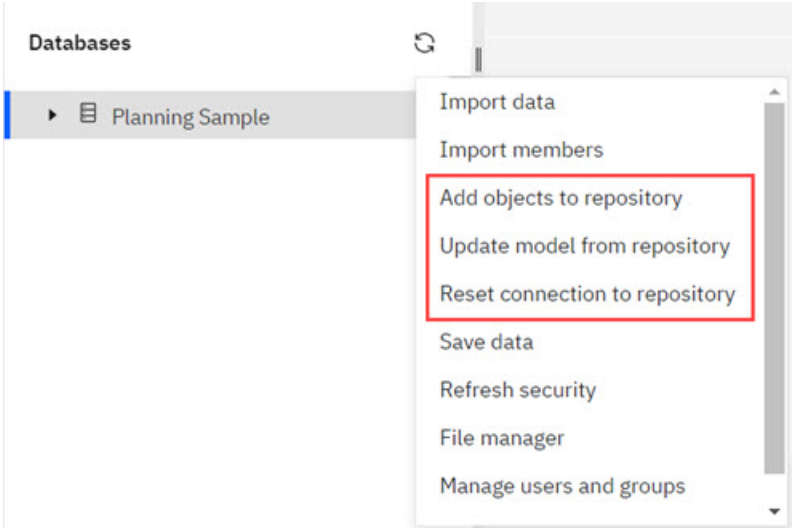
The following new and changed modeling features are available in Planning Analytics Workspace 2.0.86.

## Git repository integration for Planning Analytics database objects

Modelers can now access a Git repository (GitHub, Microsoft Azure, or AWS CodeCommit) directly from Planning Analytics Workspace. You can use the Git repository to store Planning Analytics database objects such as cubes, dimensions, rules, and processes that are common across multiple models. You can also use the repository to move database objects from one environment to another. For example, you might want to copy database objects from a development environment to a production environment.

The repository can also serve as a version control facility for your Planning Analytics database objects.

New options to interact with a Git repository are available when you right-click a database on the **Databases Tree** of a modeling workbench.




You can access a Git repository in Planning Analytics Workspace using an SSH connection and a public key/private key pair. An optional passphrase is also supported.

The following limitations apply when pushing database objects to a repository:

- Data is not included when you push cubes or views to a repository.
- Attributes are not included when you push dimensions to a repository.

For more information, see Use Git repository integration to manage Planning Analytics database objects.

## Dimension editor search and locate in hierarchy

You can determine a member's location using the dimension editor search function.

The dimension editor search function includes a menu item called **Locate**. Searching in the dimension editor acts as a filter to generate a list of members that match the search term. After searching, right-click a member from the generated list and select **Locate** to remove the filters and locate the member in the hierarchy.

**Note:** If the user picks a set (filter), the element is located within the set.

To learn more about the dimension editor, see Dimensions in Planning Analytics Workspace.

## Hide databases in Preferences

You can now use a Preferences option on a modeling workbench to hide databases from the Databases tree. When you hide a database, it is hidden in the Databases tree on both the modeling workbench and

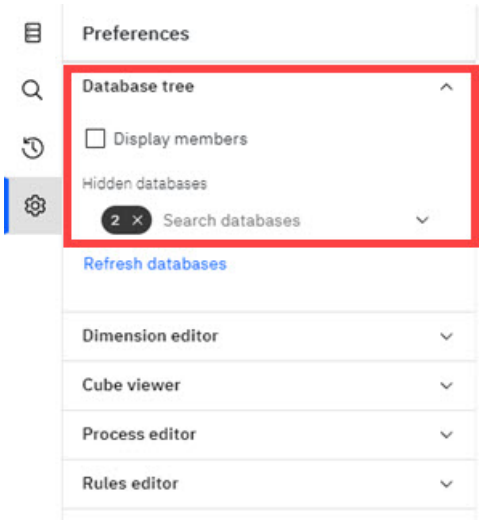
in books (Reports and Analysis). Though hidden, an active database remains running and available to processes, chores, action buttons, and any other components or features that reference the database.

1. Open a modeling workbench.
2. Click to open **Preferences.**
3. Click the **Database tree** preference section.
4. On the **Hidden databases** preference, search for the database that you want to hide or click the **Open** icon to reveal all running databases.

   If there are fewer than five databases running in your environment, you don't need to click the **Open** icon; all running databases are shown

   The list of running databases is static, so it's possible that a database could start after you open the **Preferences** and not be visible in the list. You can click **Refresh databases** to update the list.
5. Toggle the visibility of a database by clicking the box next to its name.

   The **Hidden databases** preference indicates how many databases are hidden.




6. To see your changes, refresh the **Databases** tree on the workbench.

This new preference also applies to the display of databases on the search panel. There is no impact on open editors or saved workbenches, however, **Find in tree** will not find hidden databases.

**Note:** While this preference can be set only on modeling workbenches, the setting also applies for the same modeler user in books. If a modeler hides a database on the tree in a modeling workbench, the database is also hidden for the same modeler on the tree in a book.

For more information on preferences, see Workbench preferences.

## Upcoming change notifications - 2.0.86

The following changes are planned for an upcoming Planning Analytics Workspace release. These notifications are made available so you can plan upgrades and maintain your Planning Analytics Workspace environment.

### Upcoming configuration modification required for Learn Pane in Planning Analytics Workspace Local

The cognitive Learn Pane in Planning Analytics Workspace Local will be hosted on a new domain in the near future. This will require configuration changes to Planning Analytics Workspace Local installations to ensure continued access to documentation and other information through the Learn Pane.

**Important:** This notification applies **only** to Planning Analytics Workspace Local. The change in hosting domain will not interrupt access to content through the Learn Pane for Planning Analytics Workspace on Cloud, Planning Analytics on Cloud Pak for Data, or Planning Analytics as a Service.

Full details on the configuration modifications required for Planning Analytics Workspace Local will be provided in the Planning Analytics Workspace 2.0.87 new features documentation.

## 2.0.85 - What's new, March 22, 2023

Learn about new features and known issues in version 2.0.85 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

## What's new in books and reporting - 2.0.85

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.85.

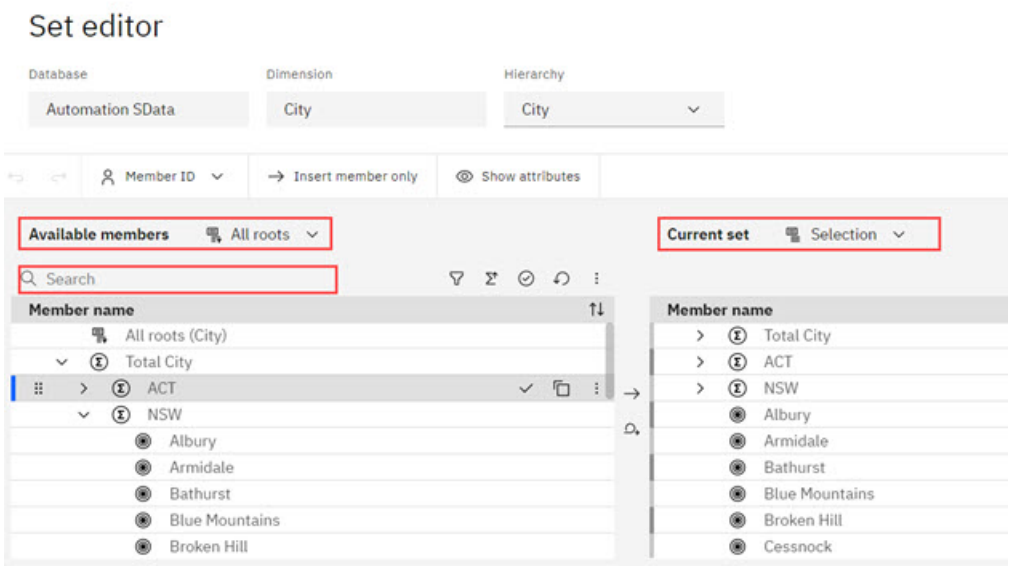
### Set editor usability improvements

Usability improvements in the set editor make it easier to work with sets.

These changes do not introduce new functionality. Rather, some options on the set editor have been moved or made more intuitive.

- The **Available members** menu now appears immediately adjacent to the **Available members** label, rather than beneath the label.
- The **Current set** menu now appears above the current set, rather than on the header.
- The search bar for the **Available members** pane is always visible, you no longer need to click a **Search** icon to reveal the search bar.

This image shows the new location of all the improvements.



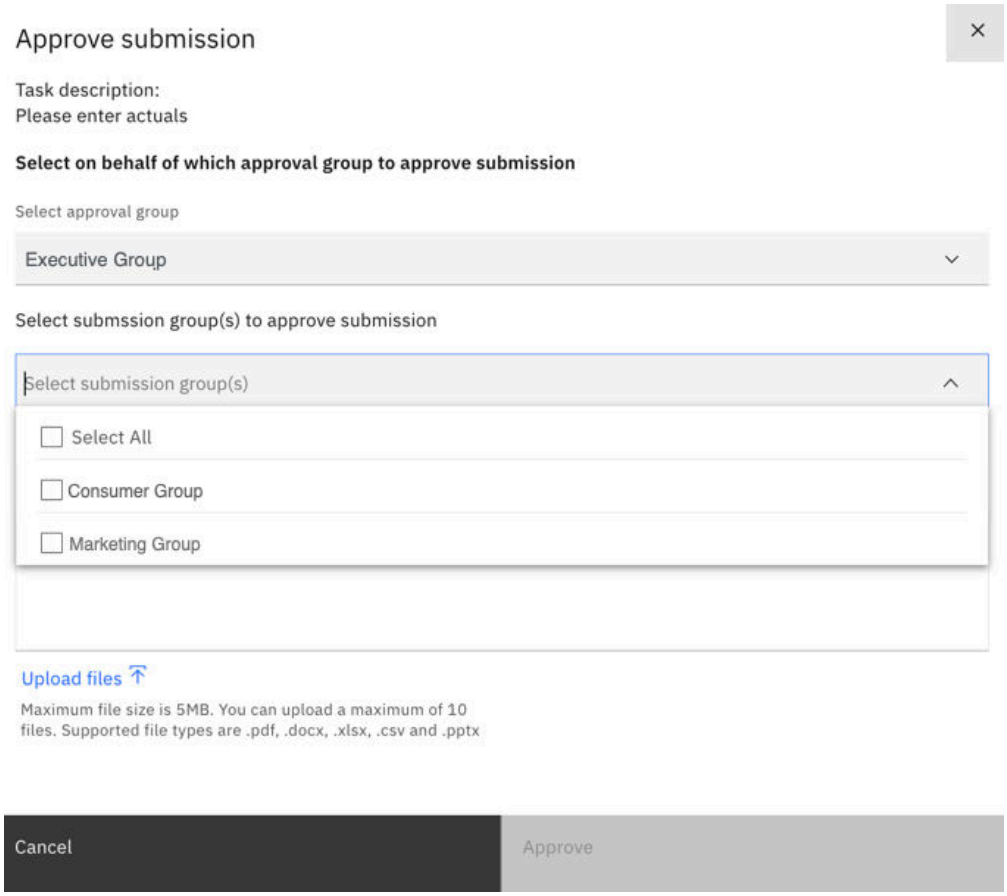

# What's new in plans and applications - 2.0.85

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.85.

## Improved approval process for plan submissions

Plan owners and approvers can now approve or reject a submission from multiple contributors in one go.

When approving or rejecting a submission, you can select the contributor(s) whose submission you want to approve or reject, or click **Select All** to apply your action to all the contributors for that submission.




To take action on a submission:

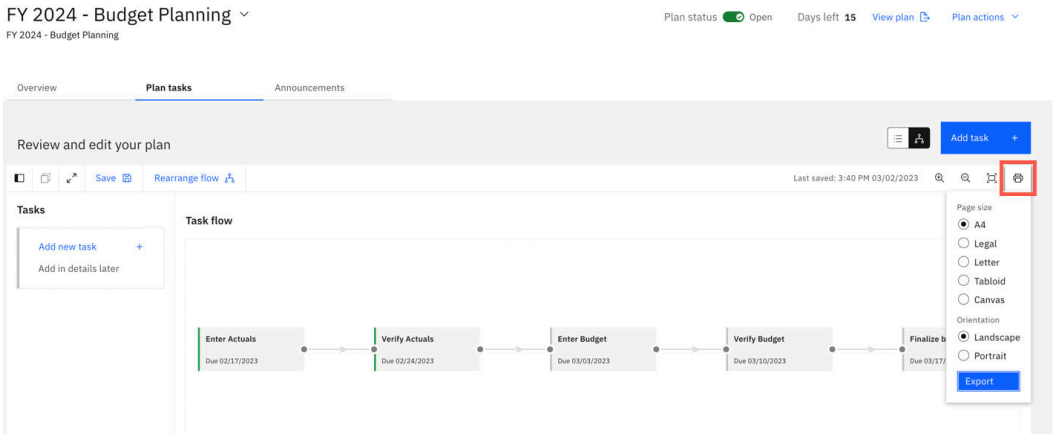
1. Click the link in the email that you receive or open the task in contribution mode.

   When a task has been submitted, approvers can see the **Reject** and **Approve** options at the end of the task, along with a history of submissions for the task.
2. Enter comments that describe your disposition on the submission, then click **Reject** or **Approve**.
3. Select the approval group that you are a part of and then select all the contributors for whom you want to approve or reject the submission. Click **Select All** to approve or reject the submission from all of the contributors for that task.
4. Click **Approve** or **Reject**.

## Save task flows in plans as PDF files

You can now save your plan's task flow as a PDF file.

In the task flow view on the **Plan tasks** tab, click the **Export** icon to save the task flow as a PDF.




# What's new in modeling - 2.0.85

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.85.

## Reopen recently opened objects from any database

You can now reopen recently opened objects from any database.

The **Recents** list contains the last twenty objects that were opened, with the most recent listed first. To remove an object from the **Recents** list, click on the **x** next to the object name.

The file path to the object includes the database name. For example, `Planning sample > Dimensions > Account`. Since search is restricted to one database, objects in search that have no path will only have their database name displayed in **Recents**.

## Workbench preferences

Enhancements to user **Preferences** are available across multiple components.

### Design update

A few design changes have been made to **Preferences**.

The **Preferences** icon is now ⚙. This button is always visible, even when the workbench is empty..

The order of sections and their sub-sections in **Preferences** is as follows:

- **Database tree**
  - **Display members**
  - **Hide Databases**
- **Dimension editor**
  - **Show dimension editing warning**
  - **Compact header**
  - **Value bar**
  - **Attribute type**
  - **Element index**
  - **Contex menu**
  - **Page size**
- **Cube viewer**
  - **Show hierarchy name**

  - **Show context information**
  - **Show icons**
  - **Show section header**
- **Process editor**
  - **Font**
  - **Font size**
  - **Enable line wrapping**
  - **Lock generated statements**
- **Rules editor**
  - **Font**
  - **Font size**
  - **Enable line wrapping**

By default, all sections are collapsed, but retain their previous state when the user opens or closes the **Preferences** panel. Opening and closing **Preferences** also refreshes the selected values.

**Note:** The selected preferences apply only to the workbench, not the dashboard.

## Process editor preferences

A new user preference is available to lock or unlock generated statements in the process editor.

To use this feature, click **Preferences** on a workbench, then click **Process editor**. Click the **Lock generated statements** option to toggle the lock on generated statements.

When the **Lock generated statements** option is enabled, users cannot edit the process script between the `#****Begin generated statements` and `#****End generated statements` lines.

## Lock dimensions in Planning Analytics Workspace

You can now lock dimensions in Planning Analytics Workspace.

A user must have either Admin or Lock privilege for a dimension to lock the dimension.

When a user locks a dimension, only those users who have Admin rights for that dimension can modify the dimension. Even the user who locks the dimension cannot modify it, unless they have Admin rights for that dimension.

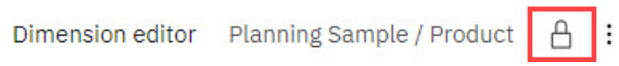
When a dimension is locked, options are disabled for these actions:

- Add members to any of the hierarchies in the dimension
- Remove members from any of the hierarchies in the dimension
- Create a hierarchy in the dimension
- Delete a hierarchy in the dimension

Users can create, delete, or update subsets under any locked dimension.

To lock a dimension, right-click the dimension on the **Databases** tree in either a book or a modeling workbench, then click **Lock dimension**. A lock icon on the dimension name indicates that the dimension is locked.

A lock icon on the header of the new experience dimension editor also indicates that the dimension is locked.




If a dimension is locked while it is open in the new experience dimension editor, you must refresh the dimension editor to see the lock icon.

To unlock a dimension, right-click the dimension on the **Databases** tree in either a book or a modeling workbench, then click **Unlock dimension**.

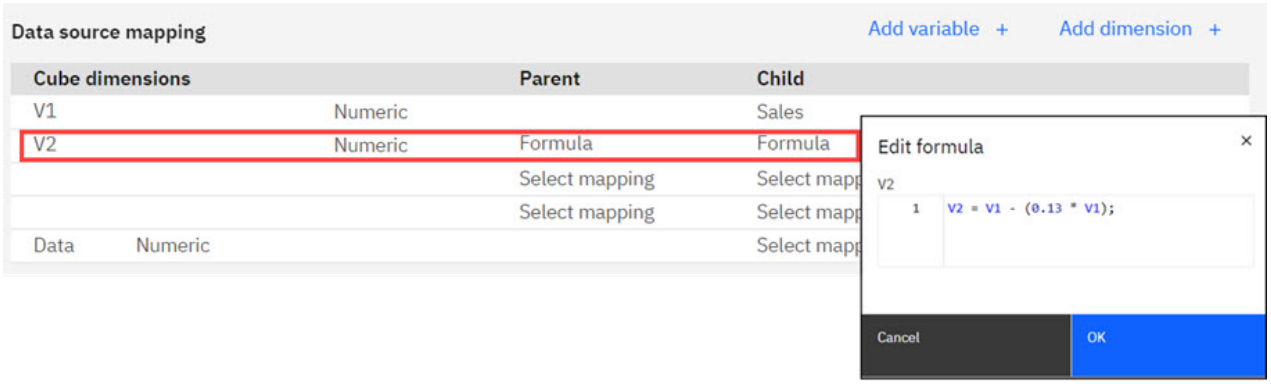
## Use formulas to manipulate a data source during a .csv import

When you use the **Import data** or **Import members** options to import from a .csv file into a workbench, you can define formulas to manipulate the source data.

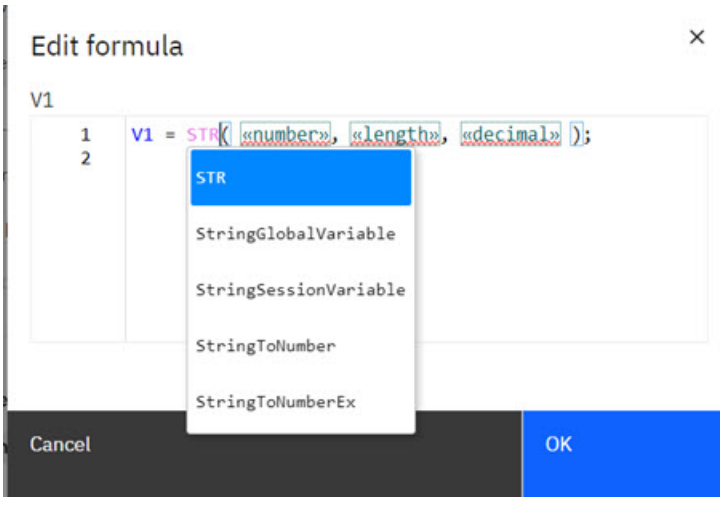
You can use any of these options on the **Databases** tree to import a .csv data source into a modeling workbench:

- Database > Import data
- Database > Import members
- Cubes node > Import data
- Cube name > Import data
- Dimensions node > Import members
- Dimension name > Import members

When mapping your data source for any of these import methods, a **Formula** mapping option is now available when you create a variable. When you select the **Formula** option, a formula editor opens and you can enter a formula to manipulate the data in the source column corresponding to the variable.

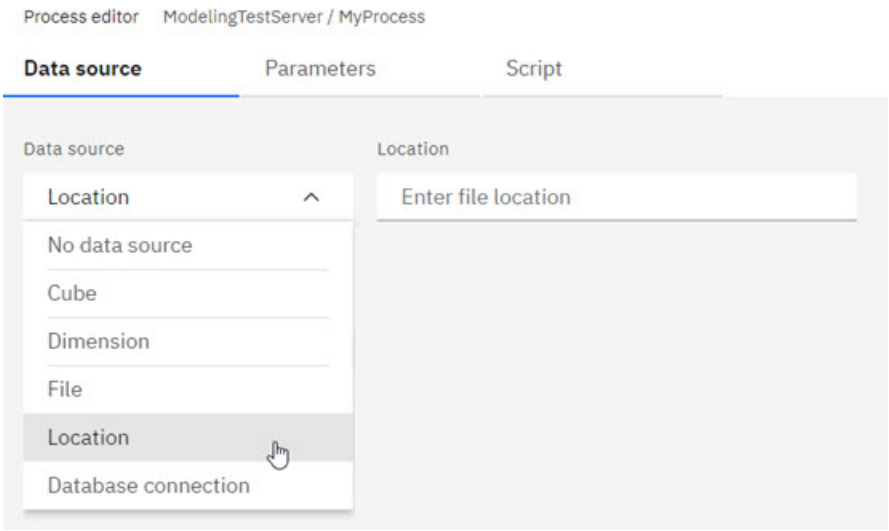



You can create a formula to perform an arithmetic calculation, manipulate strings, or apply any available Planning Analytics function to your data source. The formula editor includes an auto-complete feature to help you select and complete applicable functions.



### New 'Location' data source option for processes

You can now specify a local or network location for a TurboIntegrator process data source.




When you select the **Location** data source option, the **Location** box accepts a local or network location for your data source. For example, `s:/abc/datasource.txt`.

For more information, see Define a location data source.

## Known issues - 2.0.85

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

### Attachments in applications or plans are permanently deleted when you delete a snapshot in Lifecycle Management

If you delete a snapshot that contains an application or plan, any attachments are permanently deleted from the application or plan.

After you delete the snapshot in Lifecycle Management, the attachments can no longer be opened from the application or plan.

This issue is resolved in Planning Analytics Workspace 2.0.86.

# 2.0.84 - What's new, February 16, 2023

Learn about new features and known issues in version 2.0.84 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

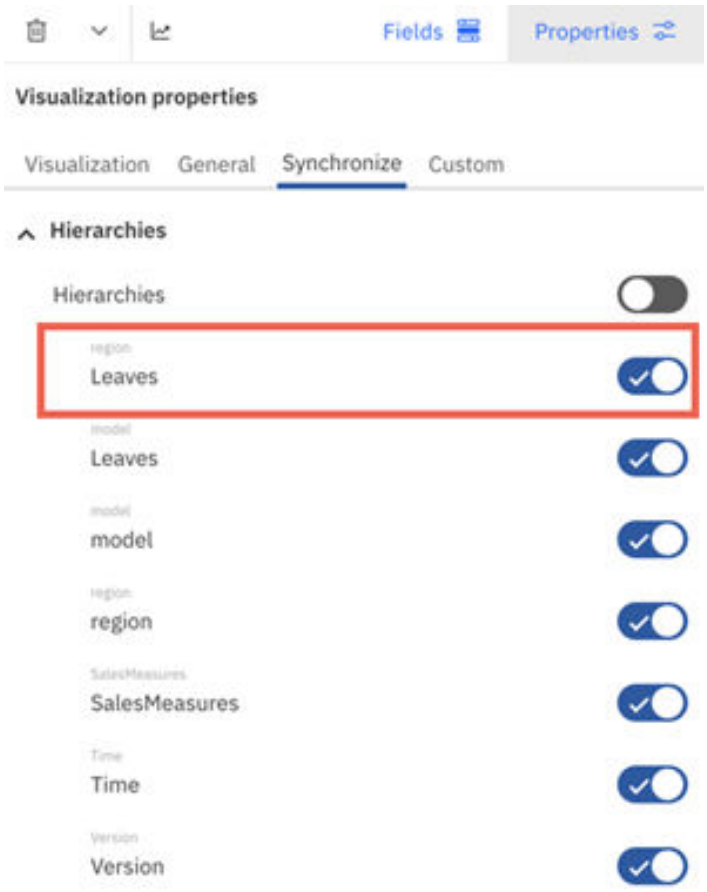
# What's new in books and reporting - 2.0.84

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.84.

## Usability improvement in synchronization for visualizations

The **Synchronize** feature for visualizations was improved for usability in Planning Analytics Workspace 2.0.84. Previously, the user interface displayed dimension synchronization labels instead of hierarchy synchronization.

You now see the hierarchy synchronization option in the **Synchronize** feature under **Visualization properties**. Dimension names also display above the hierarchy names now for easy identification.

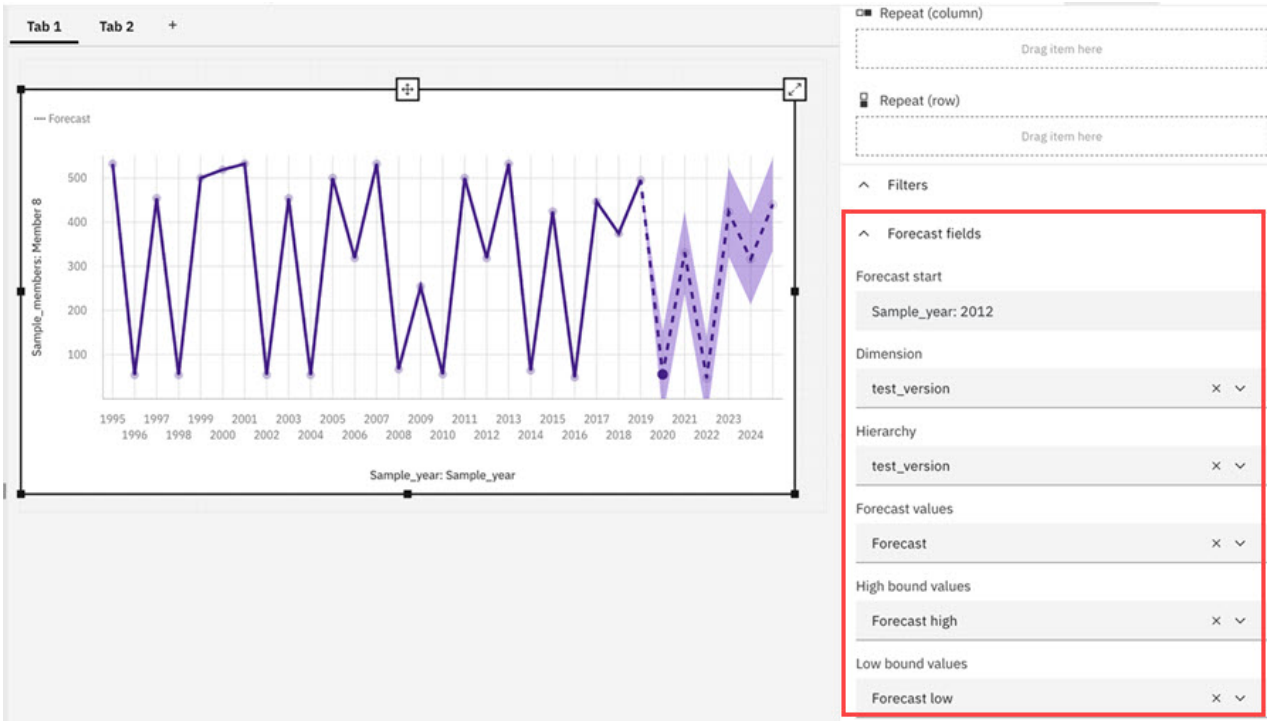



There is no other change to the **Synchronize** feature and it continues to work as before. For more information, see Synchronize objects in a book or sheet.

## New forecast capabilities for Line visualization

The Line visualization now includes three slots to specify **High bound**, **Low bound**, and **Forecast** values. You can use this new visualization to view existing forecast data independent of the forecasting interface in Planning Analytics Workspace.

As with current forecasting capabilities, **High bound** and **Low bound** are used to draw the envelope and **Forecast** indicates the forecasted values. Additionally, you must specify a **Forecast start** to indicate when the forecast data starts.

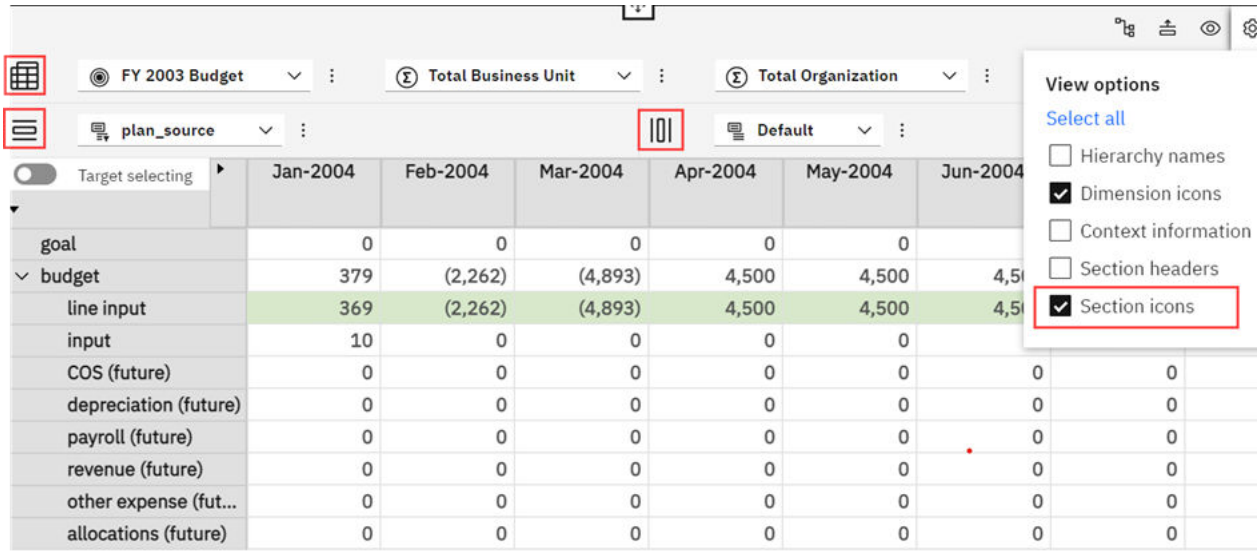



## View section icons

Section icons can now be enabled in a view independent of section header strings.

To enable or disable section icons in the cube viewer, click **View options**, and then toggle section icons by clicking the check box next to **Section icons**.

When section icons are enabled, they are displayed on the context, row, and column sections.

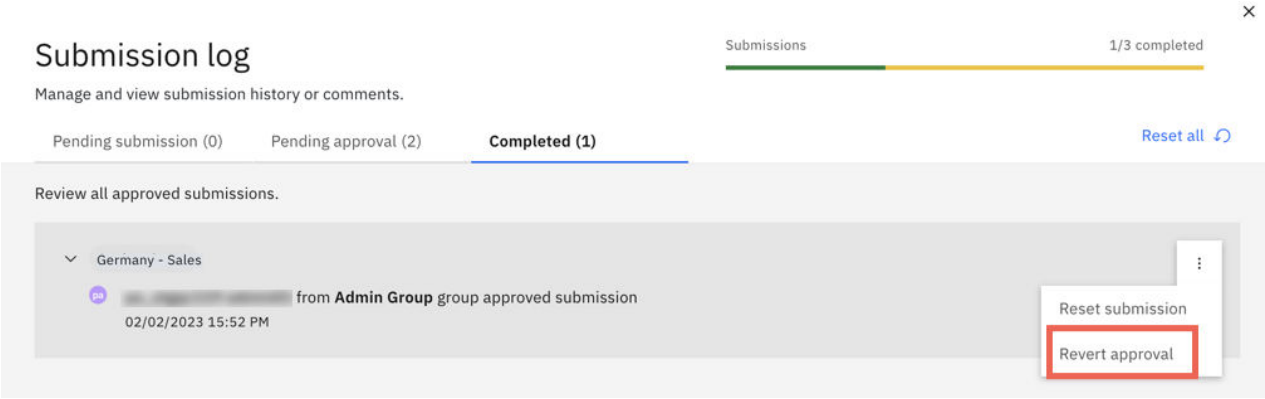



# What's new in plans and applications - 2.0.84

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.84.

## Revert approvals in plans

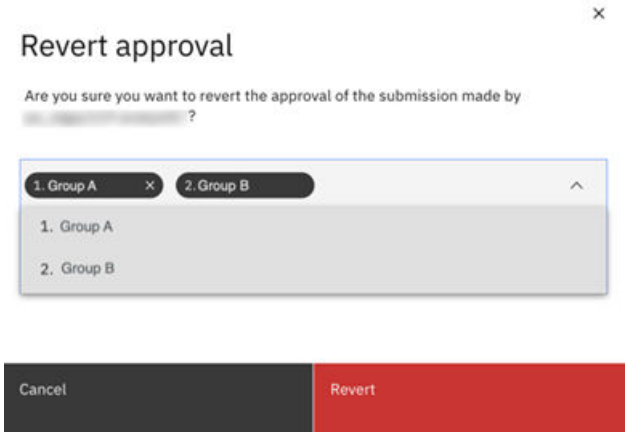
Plan owners and co-owners can now cancel previously approved tasks.




To revert a task approval:

1. Open the plan in edit mode.
2. Under **Plan tasks**, click the **Submissions** link on a task to open the **Submissions log**.
3. Click **Pending approval** or **Completed** to find the approved submission. Submissions that require multiple approvals and were only partially approved are under **Pending approval**. Submissions that were fully approved are under **Completed**.
4. Click the menu next to the task for which you want to cancel an approval and select **Revert approval**.
5. Select the approver group(s) from whom you want to remove approval(s) and click **Revert**.

   Where there are multiple approvers for a task, when you revert an approval, approval is also removed from any next-in-line approvers. In the following example, if you remove Group A's approval, Group B's is also removed. Both Group A and Group B will need to approve the submission again.




For more information, see Manage plan submissions.

## Improvements to the 'Task flow' pane for plans

Several usability improvements are introduced to the **Task flow** pane for plans in Planning Analytics Workspace 2.0.84.

The **Task flow** pane is available on the **Plan tasks** tab of a plan.

You can now edit a task directly from the **Task flow** pane. Hover the cursor on a task, then click the **Edit** icon to open the task editor.

You can also use full screen mode to maximize the canvas when working in the **Task flow** pane. Click the Full screen icon to enter full screen mode, where the **Task flow** pane and the **Tasks** list occupy the entire screen.

# What's new in modeling - 2.0.84

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.84.

## Copy object names from the Databases tree

You can now copy name objects from the **Databases** tree.

You can copy the object name into your clipboard by hovering your cursor over the item that you want to copy and then pressing C®TRL+C (Windows) or CMD+C (Mac).

When copying the object name for a member, the value copied is the MUN (Member Unique Name).

For more information, see Find data with the tree.

## TurboIntegrator process debugger

The process editor in Planning Analytics Workspace includes a debugger user interface that allows you to set line breakpoints, continue to the next breakpoint, and step over to the next line. Conditional breakpoints are also supported.

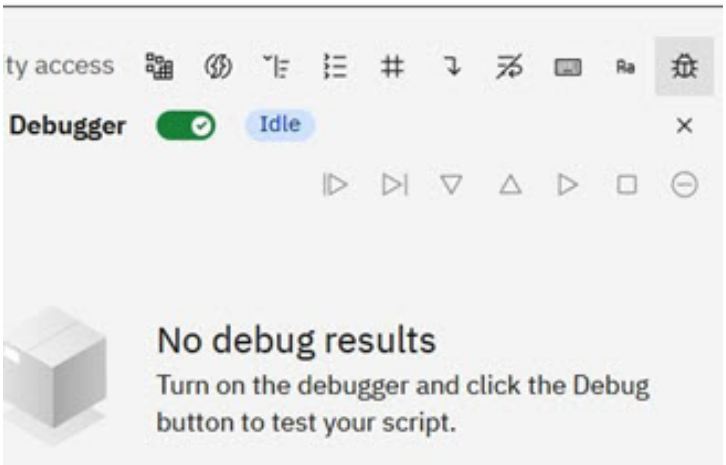
You can enable the debug mode only on one process at a time.

On the **Script** tab in the process editor, click the debugger icon to open the debugger pane.

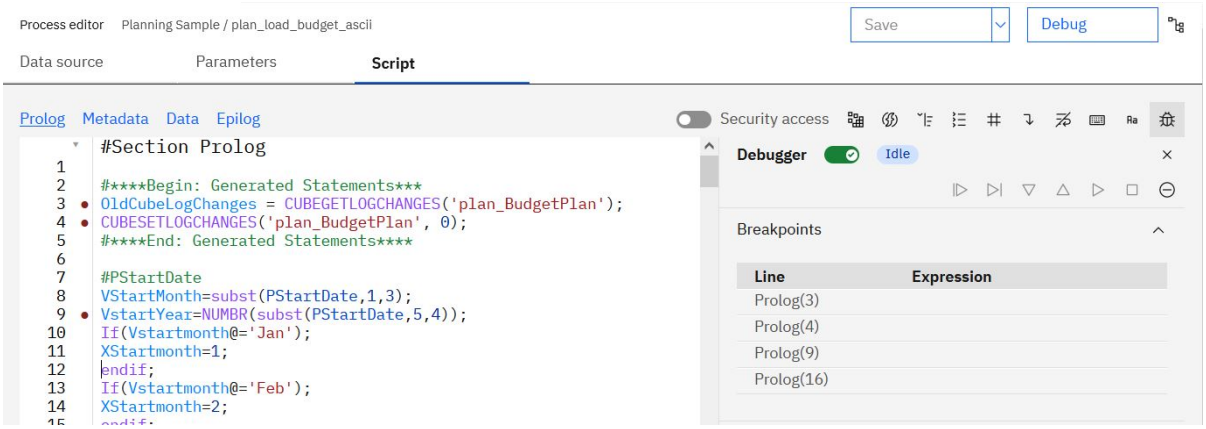



In this pane, use the **Debugger** toggle switch to enable the debug mode.




Using breakpoints, you can stop the execution of code at a particular line. To set a breakpoint, click the line number in the code editor.

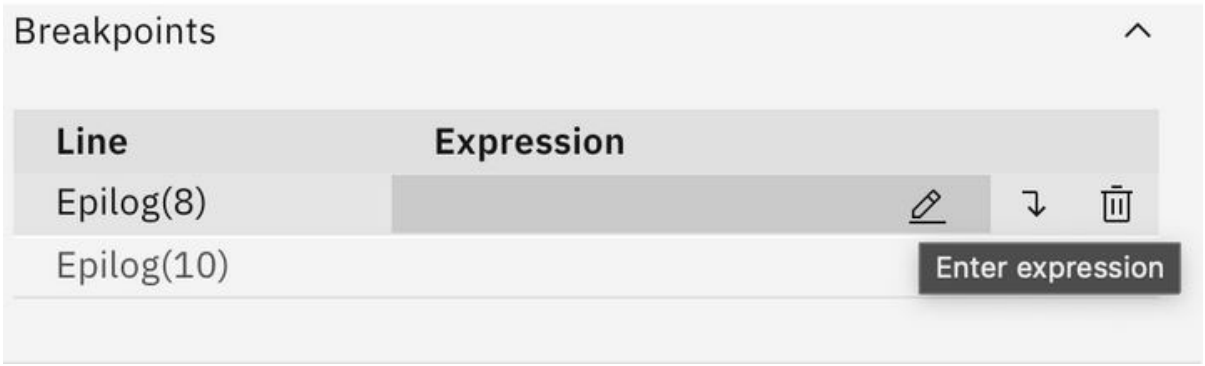



With the debugger running, navigate through the code using the following buttons:

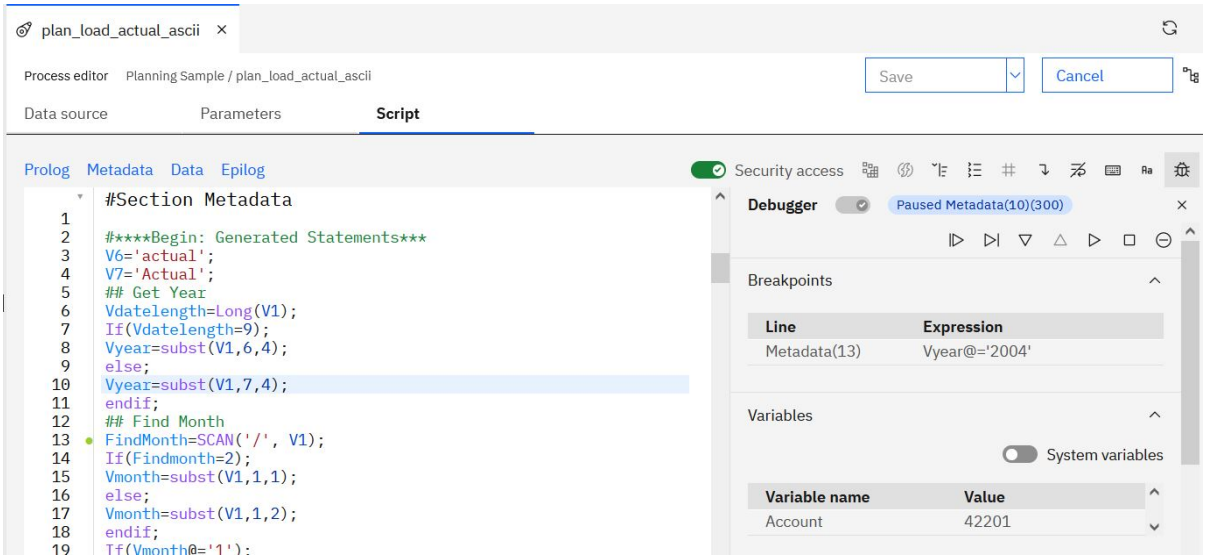
- **Continue**
- **Step over**
- **Step in**
- **Step out**
- **Continue to end**
- **Stop execution**
- **Remove all breakpoints**

You can set conditional breakpoints by entering expressions. The expression must evaluate to true or false and can contain references to any variables and parameters, and use TM1 TurboIntegrator process functions. Potential expression errors can be corrected at run time.

To enter the expression, click the **Expression** field for the selected breakpoint, and type the expression.




In the following example, the expression `Vyear@='2004'` is entered for the line 13 breakpoint to stop debugging when the input data is for year 2004.




The debugger includes a panel with the current variables, parameters, and their respective values. These values are read only.

For more information, see Debug a process.

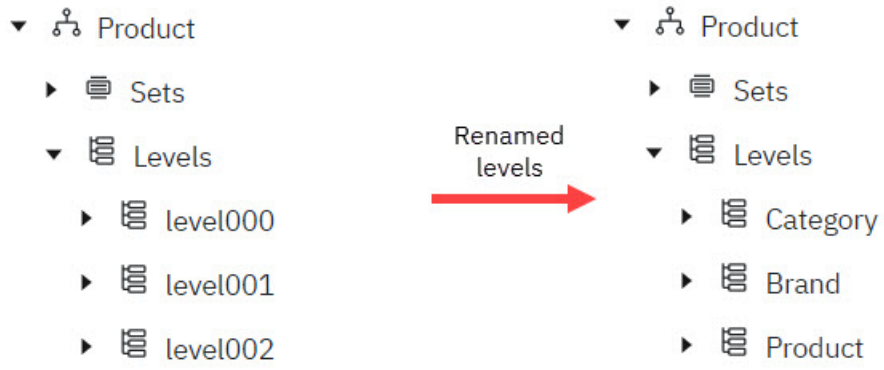
## Rename hierarchy levels in the dimension editor

You can now rename hierarchy levels directly in the new experience dimension editor.

### About this task

By default, levels within a hierarchy are named `level000` (highest level of consolidation), `level001`, `level002`, and so on.

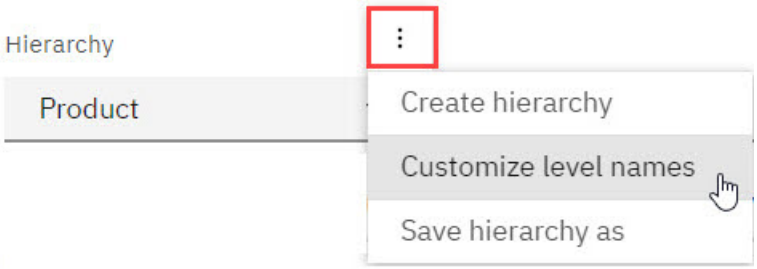
You can assign custom level names in the dimension editor to make hierarchy levels more easily identifiable. For example, the Product hierarchy has three levels corresponding to Category, Brand, and Product. Customizing the levels with the relevant names makes it easier to work in the hierarchy.




### Procedure

1. Open the hierarchy in the dimension editor.

2. Click the **Action** button next to the Hierarchy label, then click **Customize level names**.

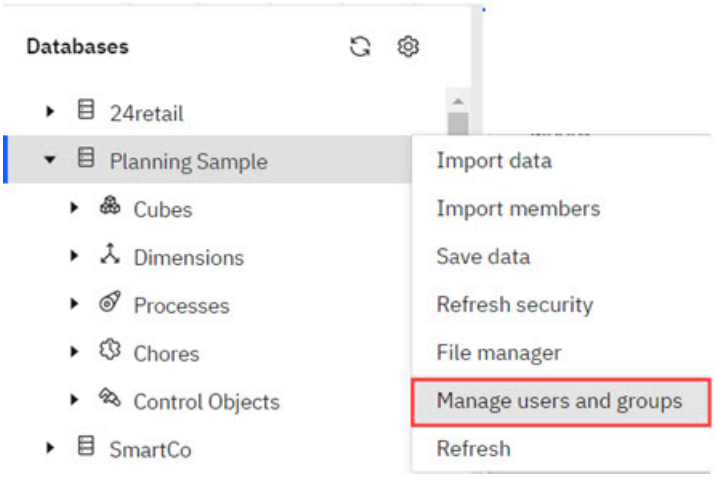



3. Enter a custom name for each hierarchy level.
4. Click **Done**.

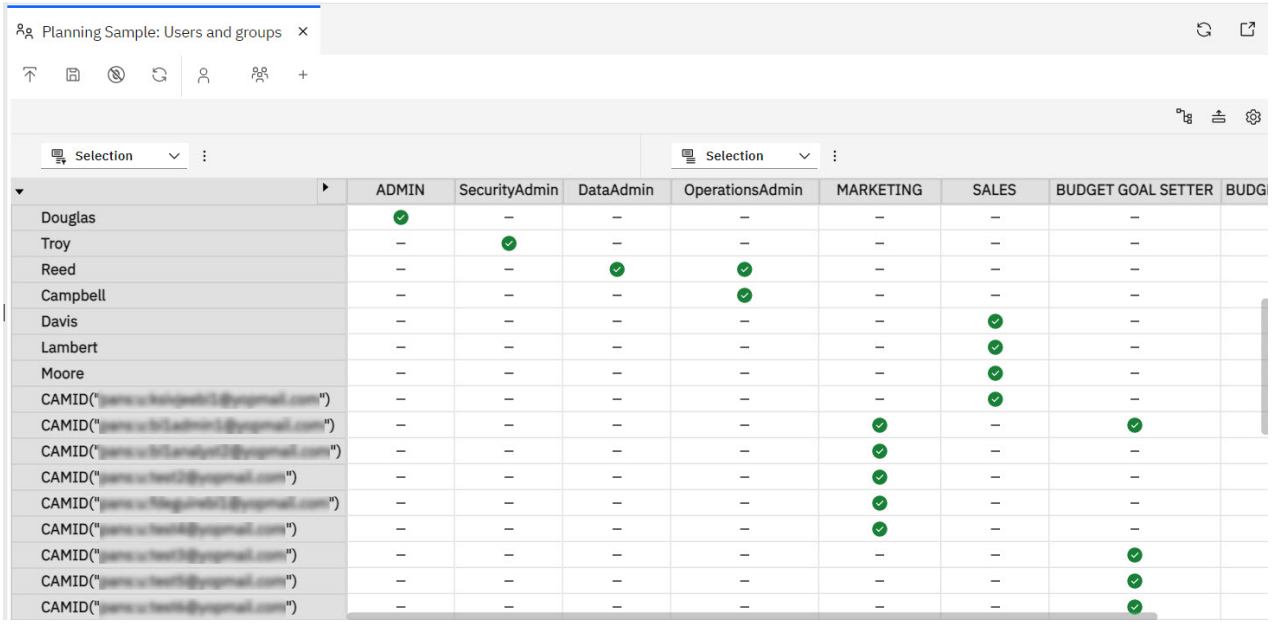
## Manage users and groups on a Planning Analytics database (Cloud only)

A new editor provides an intuitive interface to manage users and groups on a Planning Analytics database. You can also use the editor to import users and groups from Planning Analytics Workspace into a Planning Analytics database.

To open the new **Users and groups** editor on a modeling workbench, right-click a database on the **Databases** tree and then click **Manage users and groups**.




The **Users and groups** editor displays your current database users and groups in an enhanced cube view.



Use the standard view options to manage sets and to select multiple row or column headers in the **Users and groups** editor.

You can use these options to manage existing users and groups on your Planning Analytics database:

- To assign or remove a user to a group, right-click at the intersection of a user and group name, then click **Assign user to group** or **Remove user from group**.
- To assign all users to a group, right-click the group name, then click **Assign group**.
- To remove all users from a group, right-click the group name, then click **Remove group**.
- To assign a user to all groups, right-click the user name, then click **Assign user**.
- To remove a user from all groups, right-click the user name, then click **Assign user**.
- To remove a group from the database, right-click the group name, then click **Remove group**.
- To remove a user from the database, right-click the user name, then click **Remove user**.

You can also use the **Users and groups** editor to import users and groups from Planning Analytics Workspace into a Planning Analytics database or to create a new group on the Planning Analytics database.

- To create a new group on the database, click + .
- To add users that exist in Planning Analytics Workspace to the Planning Analytics database, click . Select the users, then click **Add**.

  The users are added to the database, but not assigned to any groups.
- To add groups that exist in Planning Analytics Workspace to the Planning Analytics database, click . Select the groups, then click **Add**.

  The groups are added to the database. In addition, any users within the groups are also added to the database and assigned to the appropriate groups.

# Known issues

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

## Attachments in applications or plans are permanently deleted when you delete a snapshot in Lifecycle Management

If you delete a snapshot that contains an application or plan, any attachments are permanently deleted from the application or plan.

After you delete the snapshot in Lifecycle Management, the attachments can no longer be opened from the application or plan.

This issue is resolved in Planning Analytics Workspace 2.0.86.

# Deprecation notices - 2.0.84

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.84.

### Classic dimension editor removed from the modeling workbench in Planning Analytics Workspace 2.0.84

In Planning Analytics Workspace 2.0.84, the classic dimension editor is no longer available from the modeling workbench. The classic dimension editor remains available on books.

All dimension creation and maintenance in the modeling workbench must now be performed in the new experience dimension editor. The new experience dimension editor was introduced on the modeling workbench in Planning Analytics Workspace 2.0.82.

## 2.0.83 - What's new, January 19, 2023

Learn about new features and known issues in version 2.0.83 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

## What's new in books and reporting - 2.0.83

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.83.

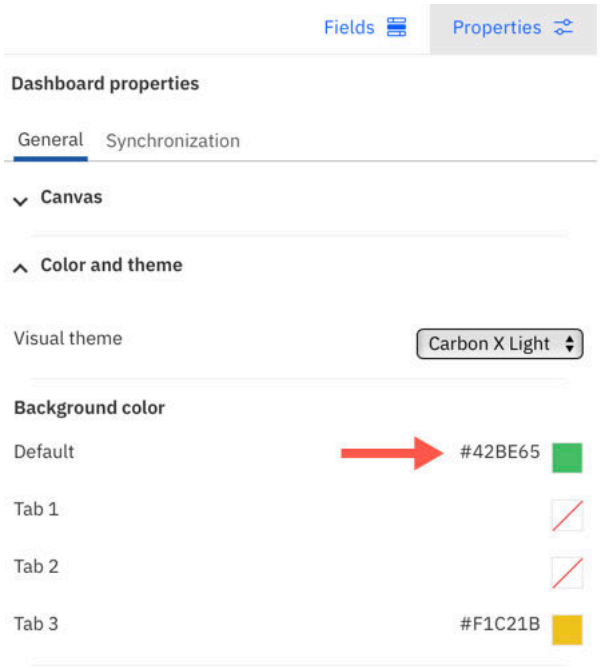
### Set default background color for tabs

You can now set a default background color for all tabs in a book. When you set a default background color, the same background color is used for any new tab that you add to the book. All existing tabs also use the default background color provided that they do not have their own color settings.

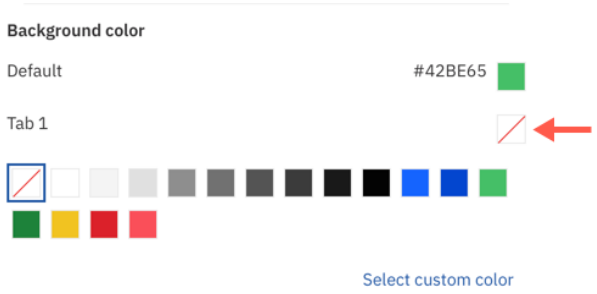
To set a default background color for tabs:

1. Click **Properties**.
2. For the entire book, ensure that nothing else has focus by clicking the background, away from any visualization or object.
3. Click **Color and theme**, and under **Background color** select a default color.

   **Tip:** You can also click **Select custom color** to open the color picker instead of choosing from the default color options.



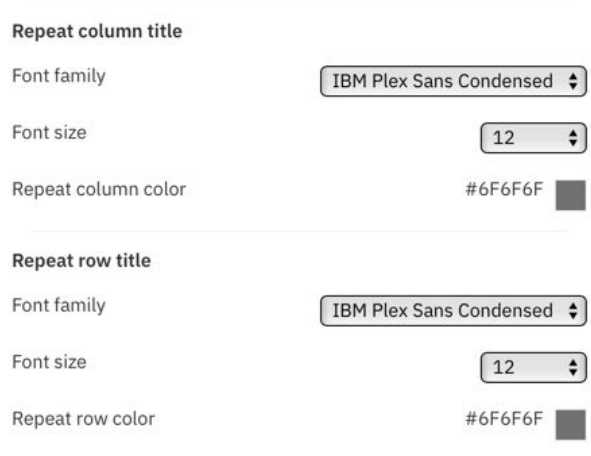

You can override the default background color for an individual tab by selecting a new color for the tab.




## Customize styling for repeat column and row titles in visualizations

You can now customize repeat column and repeat row titles in visualizations that use **Repeat (column)** or **Repeat (row)** filters.

Formatting properties for repeat column and repeat row titles appear in their own sections under **Axis** in **Properties**.



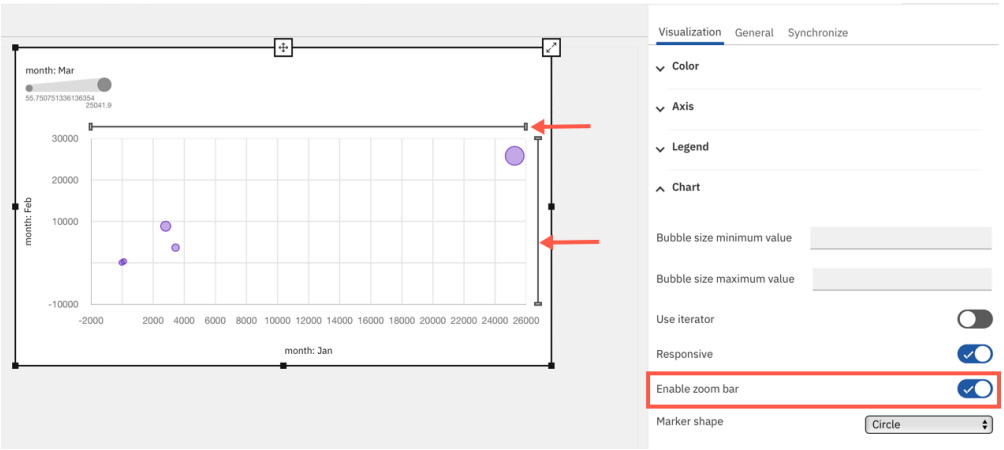
## New properties in bubble and scatter visualizations

Several new properties were introduced for bubble and scatter visualizations in IBM Planning Analytics Workspace 2.0.83:

### Enable zoom bar

The zoom bar allows for horizontal and vertical scrolling in bubble and scatter visualizations. You can now enable the zoom bar in bubble and scatter visualizations to scroll and focus on a smaller area of the visualization.

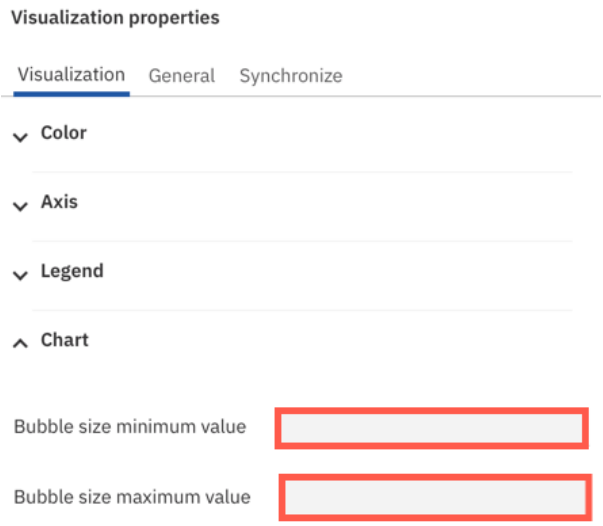
Use the **Enable zoom bar** property to add the zoom bar to your visualization. Drag a handle on the bar to zoom and focus on an area in the visualization. The area of focus changes as you drag the handle.




### Bubble size limits

You can specify bubble size limits in bubble and scatter visualizations in the **Chart** section under **Properties**.

In bubble visualizations, you can specify the minimum and maximum values for the bubble size. Bubbles for values that are below the minimum size appear as the minimum size, while bubbles for values that are above the maximum size appear as the maximum size.



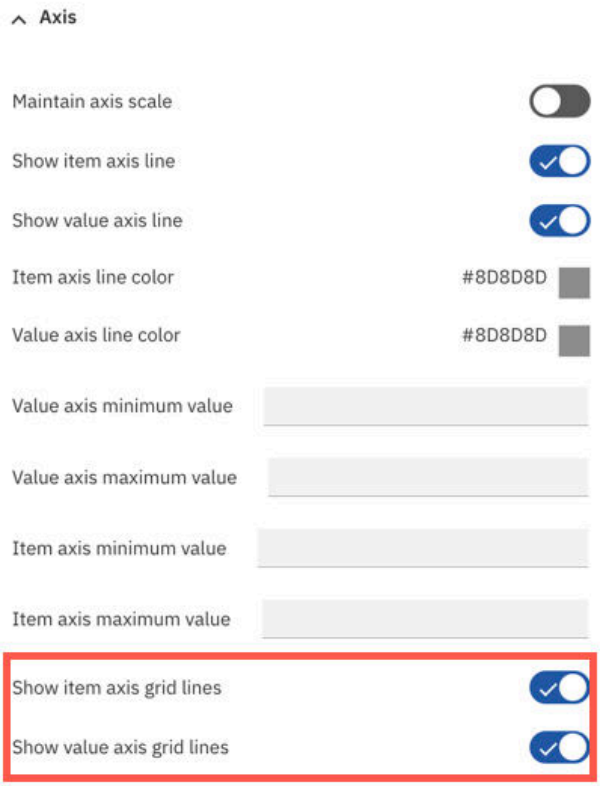

When you specify a bubble size in a scatter visualization, all points in the visualization display at this specified size. The default bubble size in scatter visualizations is **3px**.




## Show item value and axis value grid lines

Two new properties, **Show item value grid lines** and **Show item value grid lines**, have replaced the **Show grid lines** property in bubble and scatter visualizations. These new properties now let you show or hide item value and axis value grid lines independently of each other.

You can find **Show item value grid lines** and **Show item value grid lines** in the **Axis** section under **Properties**.




## New properties in line and column visualization

In IBM Planning Analytics Workspace 2.0.83, the following new properties were added in the line and column visualization:

### Enable zoom bar

The zoom bar allows for horizontal scrolling in a line and column visualization. You can now enable the zoom bar to scroll and focus on a smaller area of the visualization.

Use the **Enable zoom bar** property in the **Chart** section under **Properties** to add the zoom bar to your visualization. Drag a handle on the bar to zoom and focus on an area in the visualization. The area of focus changes as you drag the handle.

## Formatting options for column value and point value labels

You can format column value labels and point value labels in line and column visualizations. Value labels can now display as values or percentages of a category or color.

To change the format of value labels, enable **Show column value labels** or **Show point value labels** in the **Chart** section under visualization properties.

New properties were also introduced for point value labels. You can add a shadow to the labels, change their color, or set a contrast label color.

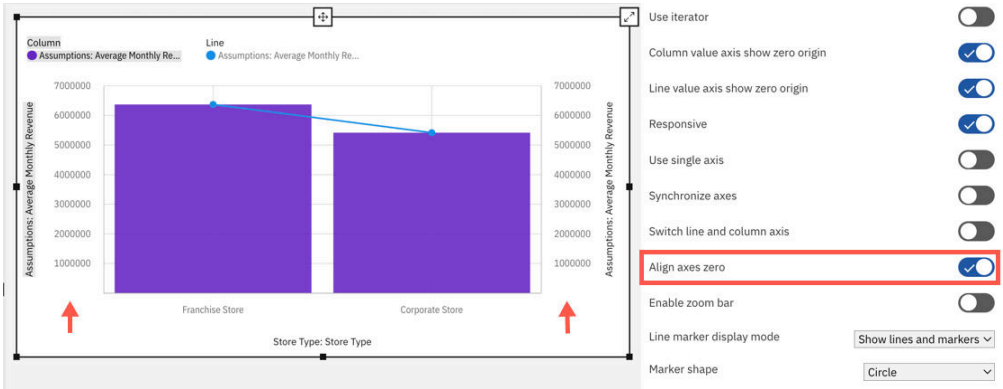



## Alignment position for zero

In line and column visualizations, you can control whether the line and column sections have zero in the same position with the new **Align axes zero** property.

Found under the **Chart** section in visualization properties, the **Align axes zero** toggle is enabled by default. If the toggle is enabled, the zero values for the lines and columns are on the same axes in the visualization. If the toggle is disabled, the zero value might be on different axes in the visualization.

Disabling the zero alignment position (that is, specifying that the line and column sections do not have zero in the same position) makes the line and column values independent and can make the relevant values more visible in certain scenarios.




## Change line width

You can now change the line width in line and column visualizations by specifying the thickness of the line. By default, the line width is set to **2 px**.




## Known issues - 2.0.83

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

### Attachments in applications or plans are permanently deleted when you delete a snapshot in Lifecycle Management

If you delete a snapshot that contains an application or plan, any attachments are permanently deleted from the application or plan.

After you delete the snapshot in Lifecycle Management, the attachments can no longer be opened from the application or plan.

This issue is resolved in Planning Analytics Workspace 2.0.86.

## Deprecation notices - 2.0.83

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.83.

### Classic dimension editor to be removed from the modeling workbench in Planning Analytics Workspace 2.0.84

As of Planning Analytics Workspace 2.0.84, the classic dimension editor will be removed from the modeling workbench. The classic dimension editor remains available on books.

All dimension creation and maintenance in the modeling workbench must be performed in the new experience dimension editor. The new experience dimension editor was introduced on the modeling workbench in Planning Analytics Workspace 2.0.82.

# 2.0.82 - What's new, November 17, 2022

Learn about new features and known issues in version 2.0.82 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

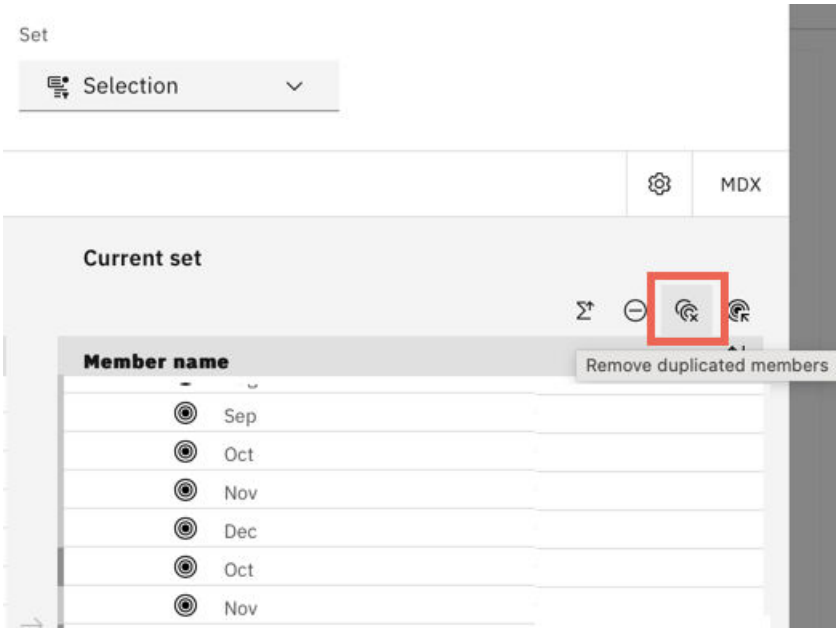
You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

# What's new in books and reporting - 2.0.82

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.82.

## Automatically remove duplicate members from current set in the set editor

You can now automatically remove duplicate members from the **Current set** in the set editor by clicking **Remove duplicated members**. This is a quick way to remove any duplicates when you are creating a dynamic or static set, especially if the set includes a large number of members.




Previously, you had to remove duplicate members by either editing the MDX expression or by selecting each duplicate member in the **Current set** and manually removing it.

## Improved error handling when visualization has a missing mapped member

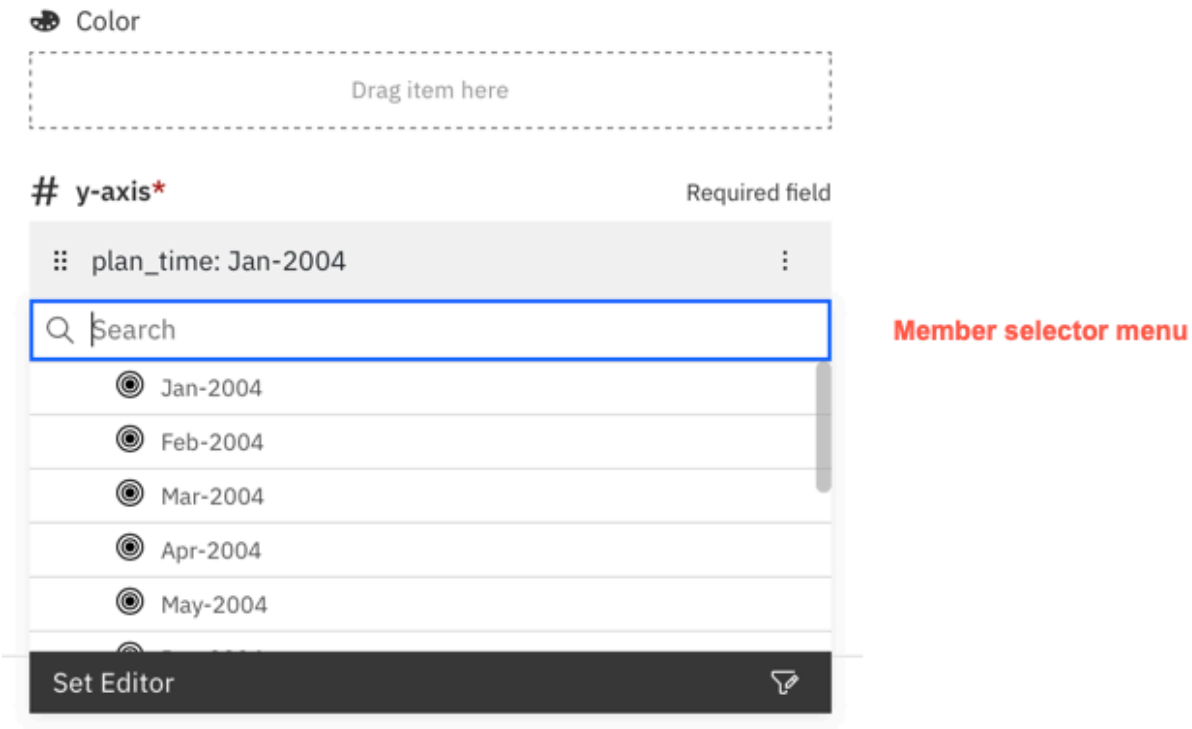
Prior to IBM Planning Analytics Workspace 2.0.82, when a visualization did not find a mapped member, an error displayed and the application showed a blank visualization. This error could occur if a mapped member was removed from the data as a result of a change in the set, zero suppression, or calculation removal, or even if the user had insufficient TM1 permissions.

In Planning Analytics Workspace 2.0.82, when a visualization cannot find a mapped member, the first member in the current set is now automatically reassigned to the field slot and the visualization displays.

You can select the visualization and go to the **Fields** tab to see the substitute member that was assigned to the missing member's field slot.

You will still see a blank visualization if:

- a substitute member is not available
- synchronization forces the visualization to use the missing mapped member
- the field slot is reassigned to a substitute member but you force the visualization to use the missing mapped member by selecting it from the member selector menu

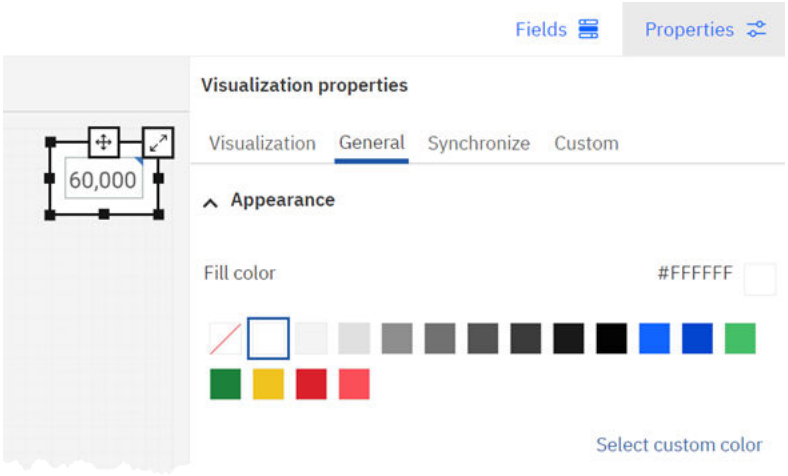



### Text fill color property added to single cell widgets

You can now change the **Fill color** of **Single Cell** widgets.

**Fill color** is now available under a **Single Cell**'s text property.

To access this feature, select the single cell visualization and go to the **Properties** tab. Under the **General** tab, click **Appearance** to see the **Fill color** options.




## What's new in plans and applications - 2.0.82

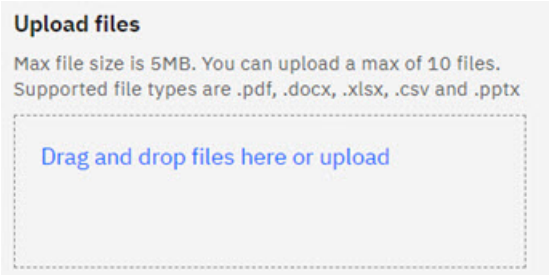
The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.82.

## Add attachments to applications and plans

You can now add attachments to applications and plans. Attachments can be added to the application or plan overview, as well as to individual tasks or sections and to announcements.

Supported file types for attachments are .pdf, .docx, .xlsx, .csv, and .pptx.

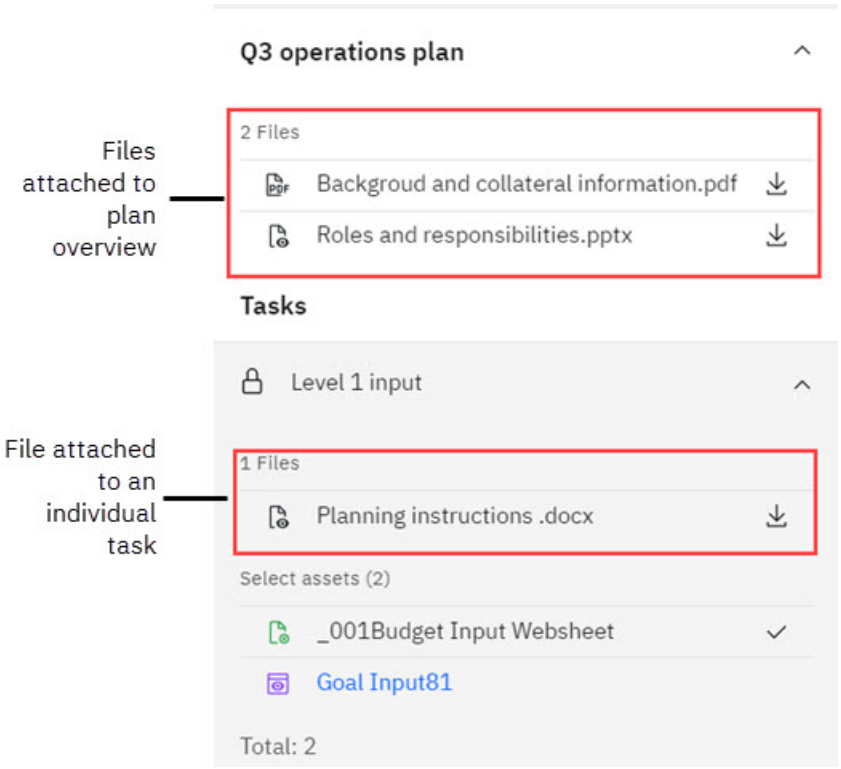
To add attachments that are accessible from the application or plan overview, use the **Upload files** option on the application or plan Details page.




To add attachments that are accessible from an individual task or section, use the **Upload files** option on the task or section Details page.

To add attachments that are accessible from an announcement, use the **Upload files** option on the Announcements page.

When a user opens a plan or application, the attached documents can be downloaded from the relevant area of the plan or application.




# What's new in modeling - 2.0.82

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.82.

## New dimension editor available on the modeling workbench

As part of IBM's continued investment in the Carbon Design methodology and in the interest of providing a consistent user experience across all components, the dimension editor in Planning Analytics Workspace has been updated to use Carbon Design principles. The new dimension editor also includes several usability and performance enhancements.

In Planning Analytics Workspace 2.0.82, the new dimension editor is available only on a modeling workbench. A future release will include the ability to use the new dimension editor on a book as well.

To use the new dimension editor to create a dimension, right-click the **Dimensions** node on the **Databases** tree, then click **Create dimension**.

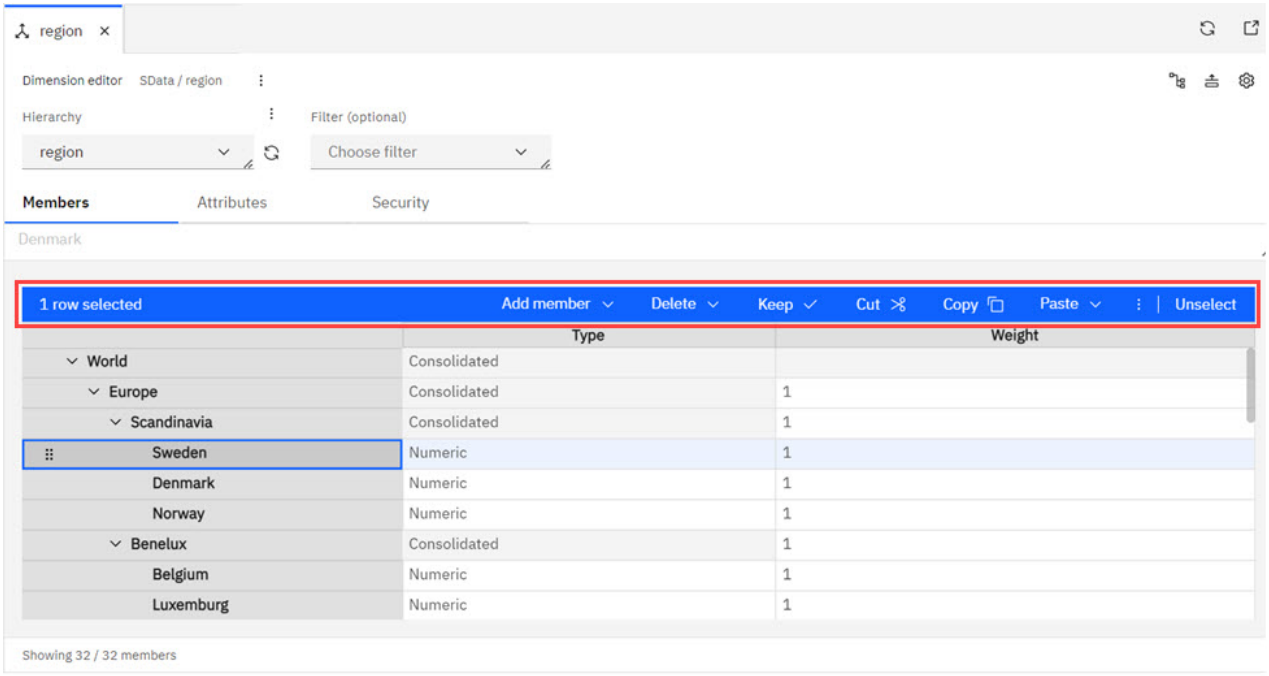
To use the new dimension editor to edit an existing dimension, right-click the dimension name on the **Databases** tree, then click **Edit dimension**.

You can also use the new dimension editor to create a new hierarchy for a dimension. Right-click the dimension name on the **Databases** tree, then click **Create hierarchy**.

The new dimension editor includes all the capabilities of the classic dimension editor, but the capabilities are provided in slightly different ways.

### Members

When you select one or more members in the dimension editor, an action toolbar becomes available.



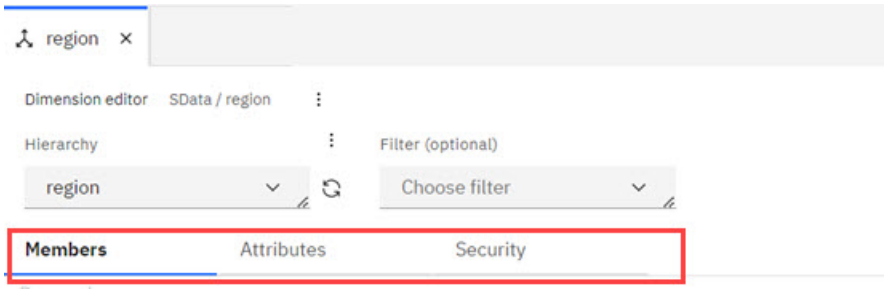

The action toolbar provides all the capabilities of the right-click (context) menu and the vertical toolbar from the classic dimension editor.

Use this toolbar to complete any of these tasks:

- Add or delete members.
- Change the order of members.
- Cut, copy, and paste members.
- Hide or reveal members.
- Set display formats.

## Attributes and security

You can create and modify member attributes and security in the new dimension editor. Click the appropriate tab to work with either **Attributes** or **Security**. These tabs take the place of the **Attributes** and **Security** icons on the classic dimension editor.

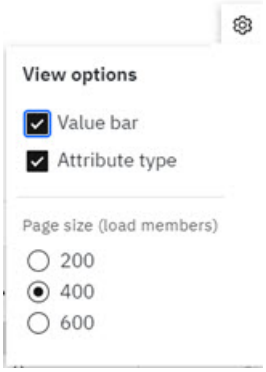



## Enhancements on the new dimension editor

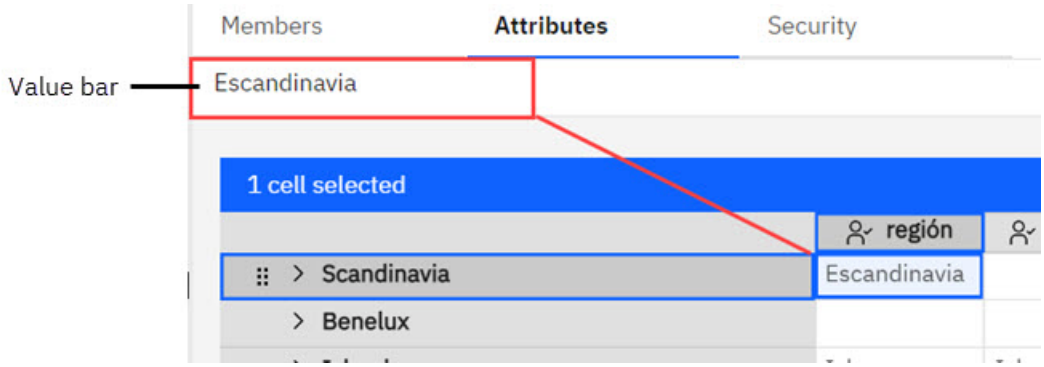
The new dimension editor includes the following enhancements.

**Settings to manage Value bar, Attribute type, and Page size**

Click the **Settings** icon to enable or disable the display of the value bar and attribute icons or to set the page size.

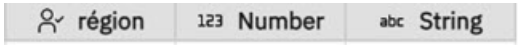



- **Value bar** - Use the value bar to edit member names or cell values outside of the dimension editor grid. This editing option is helpful, for instance, when a cell is narrow but an alias name is long.




Enable the **Value bar** option to display the value bar. Disable the **Value bar** option to maximize the number of members that are displayed in the dimension editor without scrolling.

- **Attribute type** - Enable the **Attribute type** option to display an attribute type icon for each attribute on the Attributes tab. Disable the **Attribute type** option to display only attribute names on the **Attributes** tab. When the option is enabled, you can see these icons for Alias, Number, and String icons.



- **Page size** - Select the number of members to initially load when the dimension editor is opened and each time a new page is fetched.

  You can now load and display up to 600 members per page, which is a significant performance improvement over the 70 members per page that were displayed in the classic dimension editor.

**Filter members**

Use the **Filter** option to narrow the focus of members displayed in the editor. You can select any saved set or member level.




**Expand or collapse header**

Click **Expand header** or **Collapse header** to control the display of the header.

The header displays the **Hierarchy** selector and **Filter** option. Expand the header to display both the Hierarchy selector and Filter option. Collapse the header to maximize the number of members that are displayed in the dimension editor without scrolling.

### Extend a time dimension in Planning Analytics Workspace

You might find it necessary to extend the period covered by your time dimension.

#### Procedure

1. Open the time dimension in the dimension editor.
2. Click the **Configure time** icon.
3. Select a new **End year**.
4. Click **Extend**.

#### Results

The dimension is extended to the year you select, using the granularity (months, quarters, years) you specified when you initially created the time dimension.

## Known issues - 2.0.82

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

### Attachments in applications or plans are permanently deleted when you delete a snapshot in Lifecycle Management

If you delete a snapshot that contains an application or plan, any attachments are permanently deleted from the application or plan.

After you delete the snapshot in Lifecycle Management, the attachments can no longer be opened from the application or plan.

This issue is resolved in Planning Analytics Workspace 2.0.86.

## Deprecation notices - 2.0.82

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.82.

### Classic dimension editor to be removed from the modeling workbench in Planning Analytics Workspace 2.0.84

As of Planning Analytics Workspace 2.0.84, the classic dimension editor will be removed from the modeling workbench. The classic dimension editor remains available on books.

All dimension creation and maintenance in the modeling workbench must be performed in the new experience dimension editor. The new experience dimension editor was introduced on the modeling workbench in Planning Analytics Workspace 2.0.82.

## 2.0.81 - What's new, October 20, 2022

Learn about new features and known issues in version 2.0.81 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

## What's new in Planning Analytics Workspace Local - 2.0.81

The following new and changed features are available in Planning Analytics Workspace Local 2.0.81 as of November 1, 2022.

### Configuration support available for Red Hat OpenShift Service Mesh in Planning Analytics Workspace Distributed

You can now enable Service Mesh support in IBM Planning Analytics Workspace Distributed. You can configure Planning Analytics Workspace Distributed to use Service Mesh if OpenShift Service Mesh was enabled for your cluster.

**Note:** You need to first install and configure OpenShift Service Mesh before you can enable Service Mesh in Planning Analytics Workspace Distributed.

For more information, see Configure OpenShift Service Mesh in Planning Analytics Workspace Distributed.

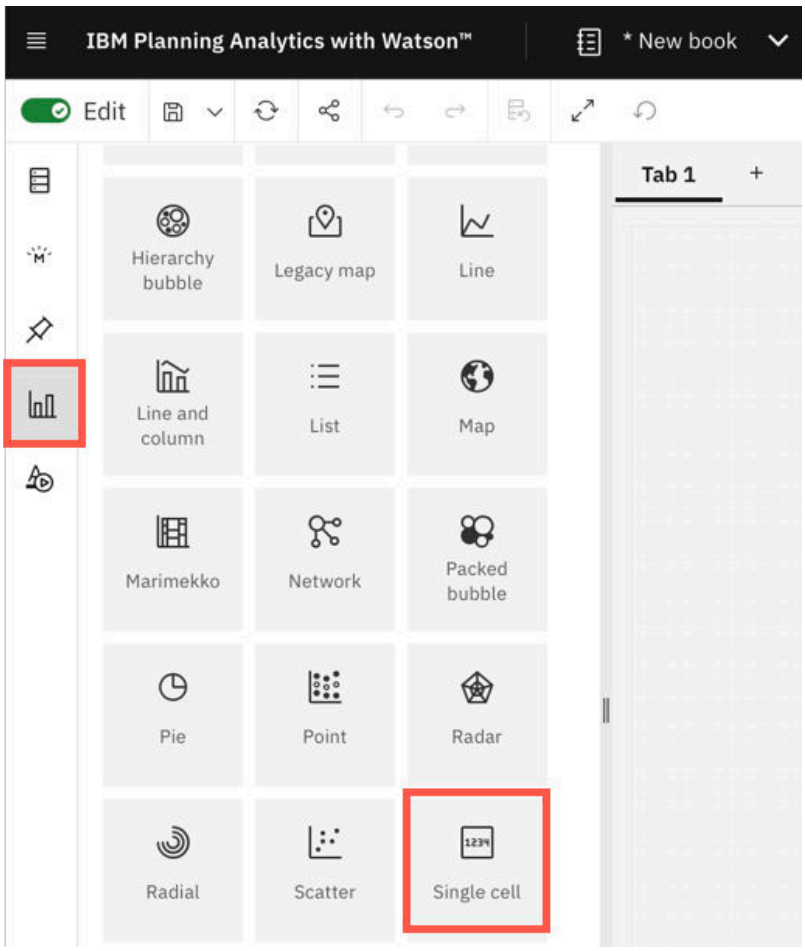
## What's new in books and reporting - 2.0.81

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.81.

### Single cell widget is now a visualization type

The single cell widget is now a visualization type and has several new capabilities. You can create a single cell visualization and convert it to a view (exploration) or another visualization and vice versa.

Like any other visualization, you can add a **Single cell** visualization to your book from **Visualizations** and drag a cube or view on to the template.




You can also create a **Single cell** visualization by right-clicking a view and selecting **Add to sheet**.

**Note:** Unless you select a specific cell in the view, when you add a **Single cell** visualization with this method, the visualization displays data from the first cell in the view.

When you select the **Single cell** visualization and go to the **Fields** tab, you can see dimensions and filters in their appropriate fields. You can also change the color palette for any conditional rules applied to the visualization and enable automatic refresh under the **Visualization** tab under **Properties**. Previously, fields and visualization properties were not available for the single cell widget.

**Note:** Any existing cell widgets in books from before Planning Analytics Workspace 2.0.81, will convert to the new single cell visualization with Field dimensions and visualization properties.

You can convert **Single cell** to another visualization or an exploration and vice versa. When you convert another visualization or view to a Single cell visualization, Single cell displays data from the first cell.

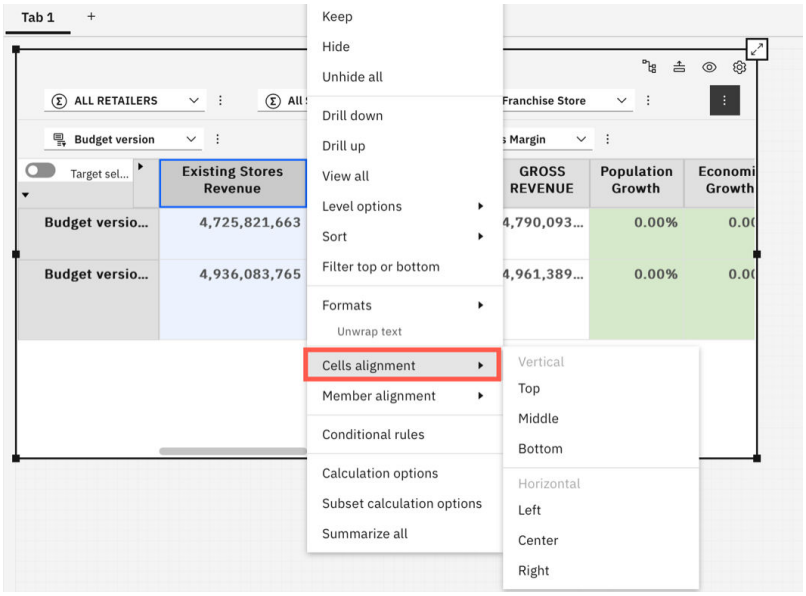
The new cell alignment option is also available for **Single cell** visualizations. Right-click a **Single cell** visualization to see cell alignment and other formatting options for the visualization.

## Align cells and headers

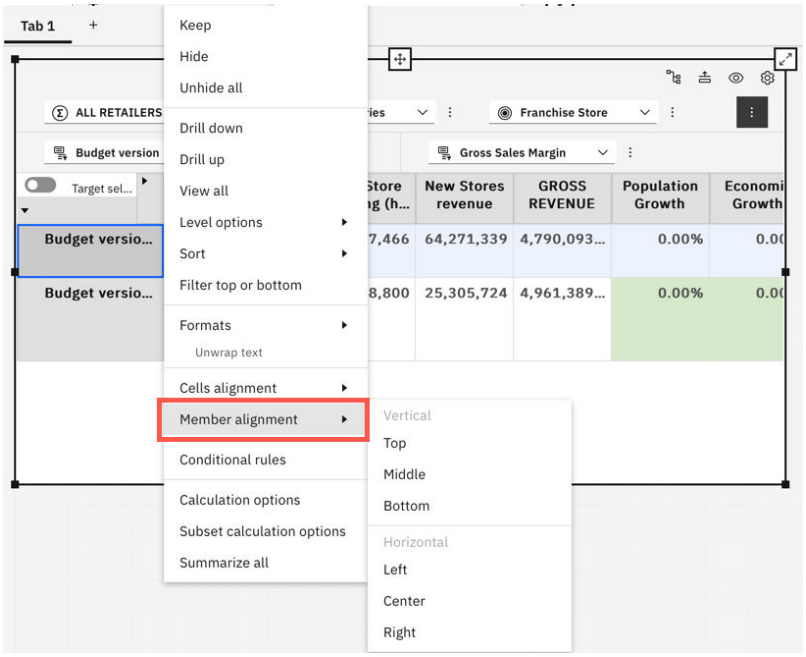
You can now align the text in data cells and headers in an exploration (view) and a single cell visualization.

In explorations, you can align text in data cells and headers either vertically or horizontally. You can align a row or column of data cells, align a member in row and column headers, and align based on hierarchy.

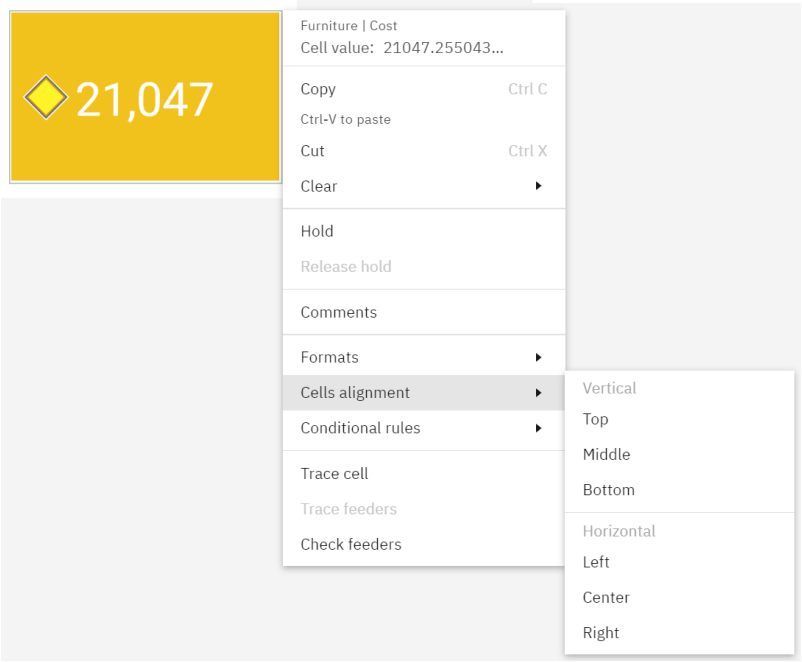
Right-click a data cell and select **Cells alignment** to see text alignment options.




To align members in row or column headers, right-click a row or column header and under **Member alignment**, select an alignment option.




You can also align text in the new single cell visualization vertically or horizontally. Simply, right-click the visualization to see alignment options.



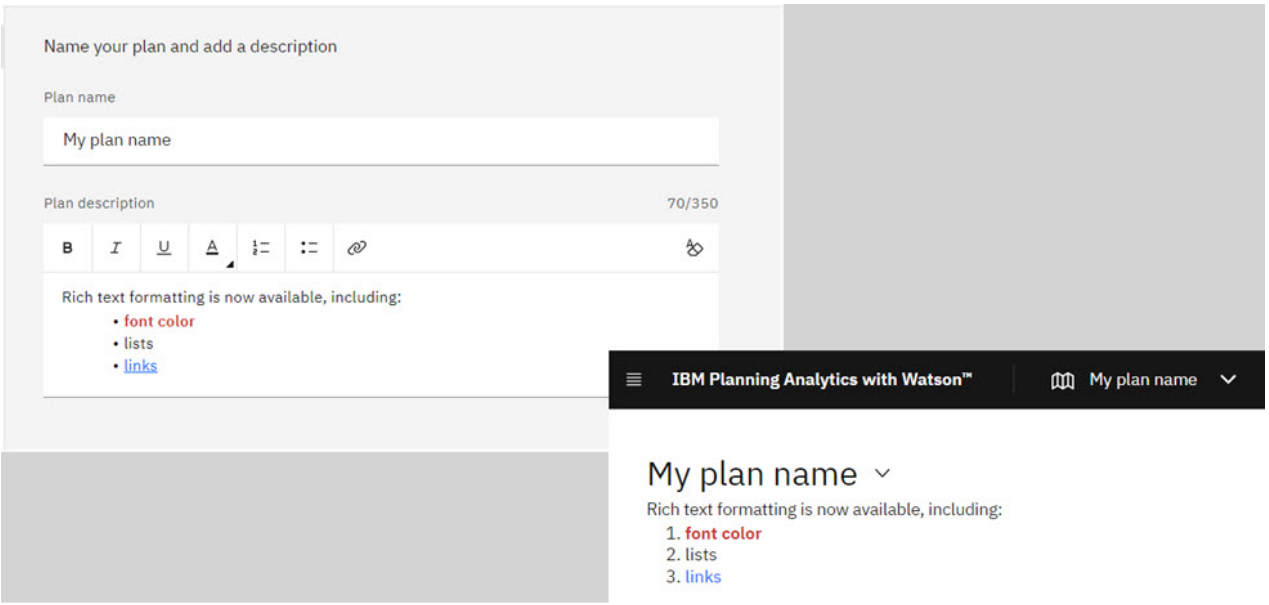

You can clear text alignment in the exploration or single cell visualization by right-clicking the cell and selecting the alignment option under **Clear**.

# What's new in plans and applications - 2.0.81

The following new and changed features in plans and applications are available in Planning Analytics Workspace 2.0.81.

## Use rich text formatting in plan and application descriptions

You can now apply rich text formatting to enhance your plan and application descriptions.



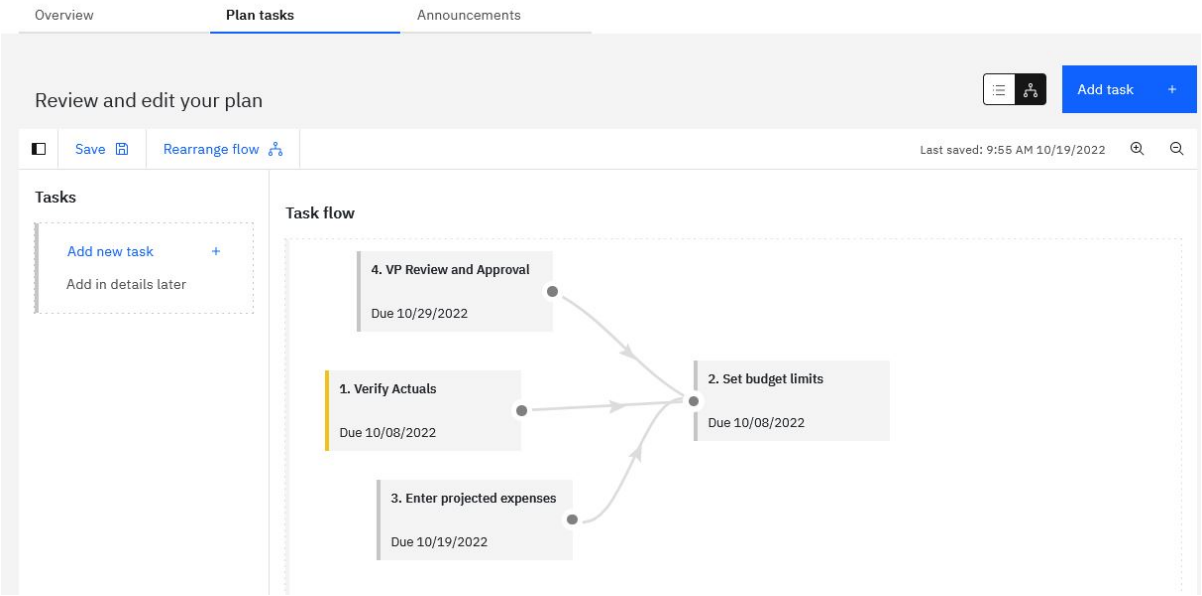

The new text formatting toolbar on the plan or application description box lets you quickly apply text formatting, create numbered and bulleted lists, and create links.

### Task flow view in plans

You can now manage plan tasks by using the task flow view.

On the **Plan tasks** tab, you can switch between the task list view and the task flow view.

In the task flow view, you can create tasks and define task dependencies. When you create a task in this view, the task is shown in the **Tasks** pane. To define the task dependencies in this view, drag the tasks from the **Tasks** pane to the **Task flow** pane. Then, hover the cursor over the task rectangle until you see a dot in the rectangle sides. Starting from the dot, draw a line toward the dependent task, linking the two tasks. A visual task flow diagram is created. You can rearrange the position of the tasks in the diagram. The diagram is saved when you save the plan.




The changes in the task flow view are reflected in the task list view, and the opposite.

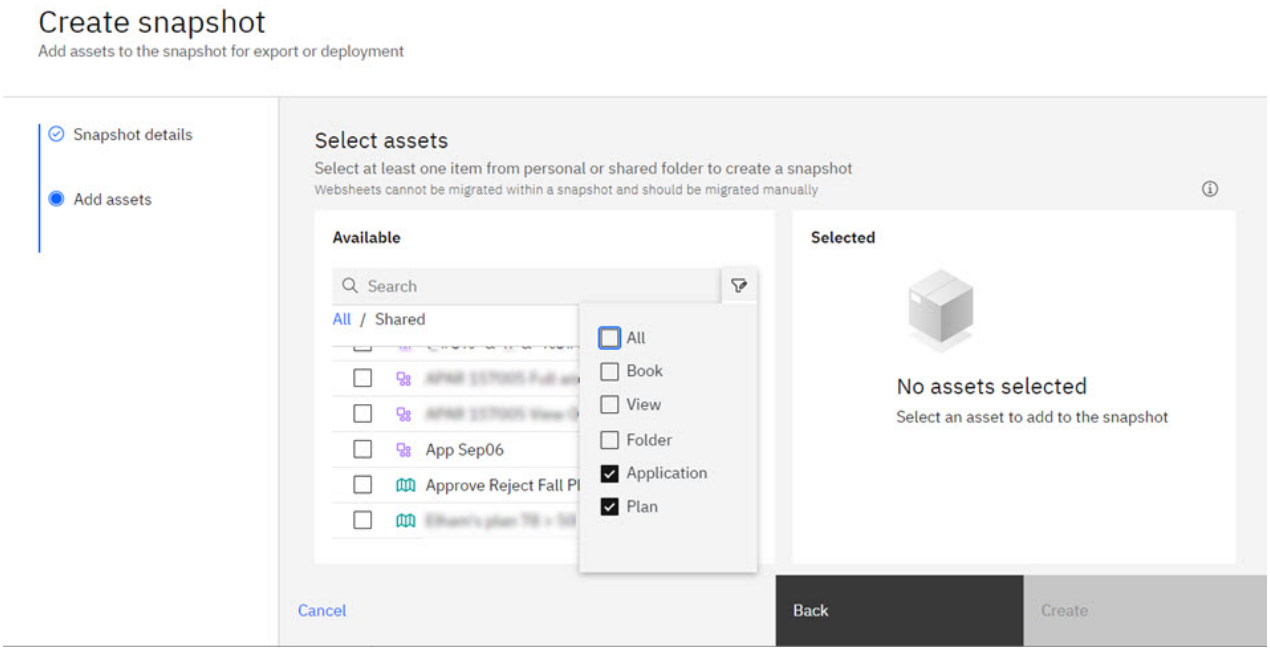
For more information, see Using the task list and task flow views.

## What's new in administration - 2.0.81

The following new and changed administration features are available in Planning Analytics Workspace 2.0.81.

### Include plans and applications when you copy and deploy assets with Lifecycle Management

You can now include plans and applications when you create and deploy a snapshot in Lifecycle Management.




When you add a plan or application to a snapshot, all books and views within the plan or application are also included in the snapshot. However, as Lifecycle Management does not support websheet migration, any websheets present in a plan or application are not included in the snapshot.

If a step in a plan includes a contributor or approver group that is not available on the target database, the group is removed from the step when the snapshot is deployed. You must manually update the step to use a valid group on the target database.

For more information about Lifecycle Management, see Copy and deploy assets with Lifecycle Management in Planning Analytics Workspace.

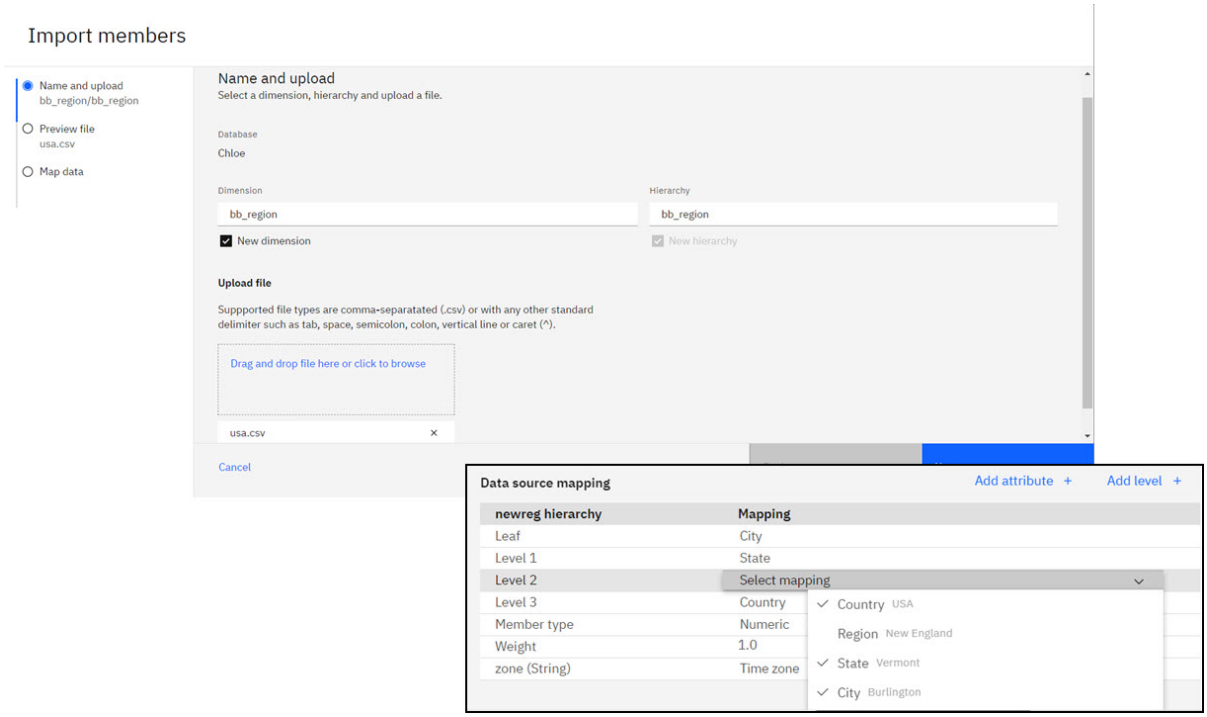
## What's new in modeling - 2.0.81

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.81.

### Updated 'Import members' procedure

As part of our continued investment in the IBM Carbon Design methodology and in the interest of providing a consistent user experience across all components, the **Import members** procedure for a dimension has been updated to use Carbon Design principles.

The new procedure uses more intuitive controls to define a data source and set mappings.




Additionally, you can import members into a dimension from multiple locations on the **Databases** tree. Depending on the launch point, your ability to create or modify metadata might be limited.

If you click **Dimensions**, **Import members** on the Databases tree, there are no limitations on metadata updates. You can:

- update an existing dimension and an existing hierarchy
- update an existing dimension and create a new hierarchy
- create a new dimension and a new hierarchy

If you click **<dimension name>**, **Import members** on the Databases tree, updates apply to the dimension you clicked, but you can pick the hierarchy to update or choose to create a new hierarchy.

If you click **<hierarchy name>**, **Import members** on the Databases tree, updates apply only to the hierarchy you clicked.

For more information, see Import members and attributes into a dimension.

## Snippets in the process editor

The process editor in Planning Analytics Workspace provides predefined code snippets that users can insert in the TurboIntegrator process.

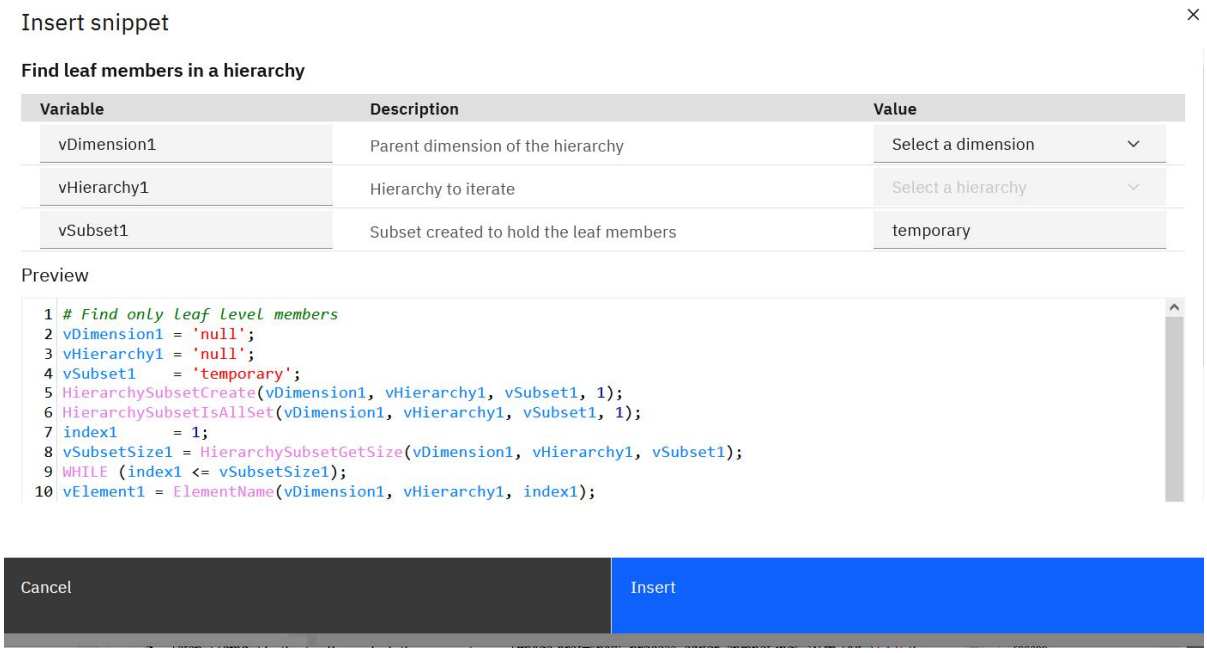
The TurboIntegrator process code can be repetitive. Users often repeat the same tasks, changing only the variable or object names. The code snippets simplify the repetitive tasks, making the process less error-prone.

To use the snippets, on the **Script** tab in the process editor, click the snippet icon in the toolbar, as highlighted in the following image:




The following categories of snippets are currently available: **Create**, **Iterators**, and **Other**. Expand the category to view the list of snippets, and select one of them.

The following image shows one of the available iterator snippets:




The **Variable** and **Value** columns are editable. As you specify values in these columns, the **Preview** is updated with your input. When you click **Insert**, the snippet is inserted in the line under the current position of the cursor.

For more information, see Insert a code snippet into a process.

# Known issues - 2.0.81

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

## Attachments in applications or plans are permanently deleted when you delete a snapshot in Lifecycle Management

If you delete a snapshot that contains an application or plan, any attachments are permanently deleted from the application or plan.

After you delete the snapshot in Lifecycle Management, the attachments can no longer be opened from the application or plan.

This issue is resolved in Planning Analytics Workspace 2.0.86.

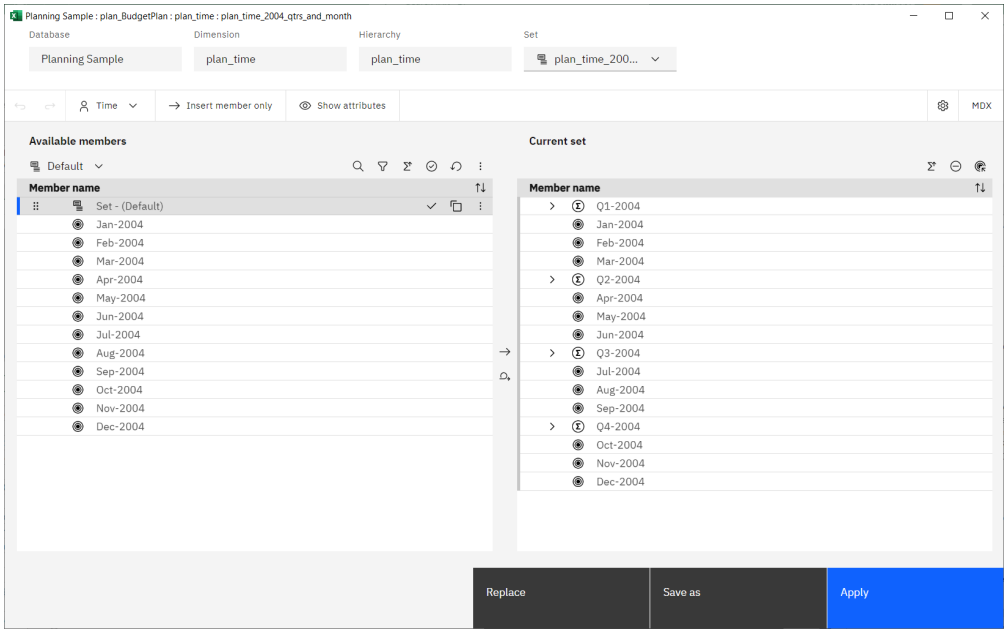
# Deprecation notices - 2.0.81

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.81.

## Deprecation notice - the set editor in a widget is removed from books as of Planning Analytics Workspace 2.0.81

You cannot add a set editor in a widget from the Data tree in books as of Planning Analytics Workspace 2.0.81.

Instead, when you edit a set from the Data tree, the new experience set editor opens. The new set editor opens as a dialog and is not inserted as a widget in the book.




**Note:** You can still add the set editor to a modeling workbench. When you open or create a set from the Databases tree in a workbench, the set editor appears on a new tab on the active tabset.

The new experience set editor was introduced in Planning Analytics Workspace 2.0.71. For more information, see New experience view and set editor.

# 2.0.80 - What's new, September 22, 2022

Learn about new features and known issues in version 2.0.80 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

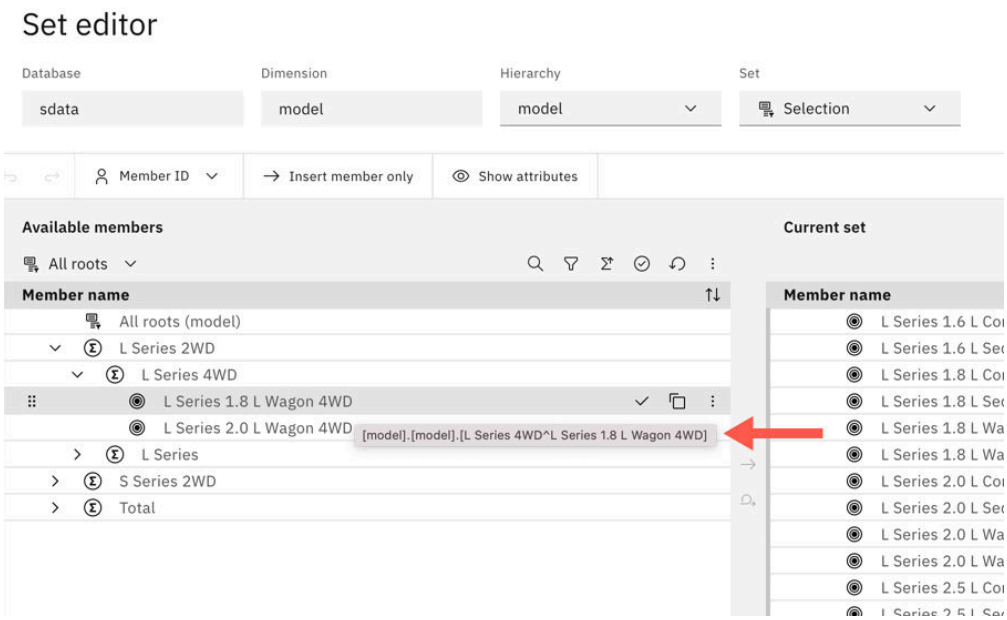
## What's new in books and reporting - 2.0.80

The following new and changed features in books and reporting are available in Planning Analytics Workspace 2.0.80.

### Member unique names in set editor

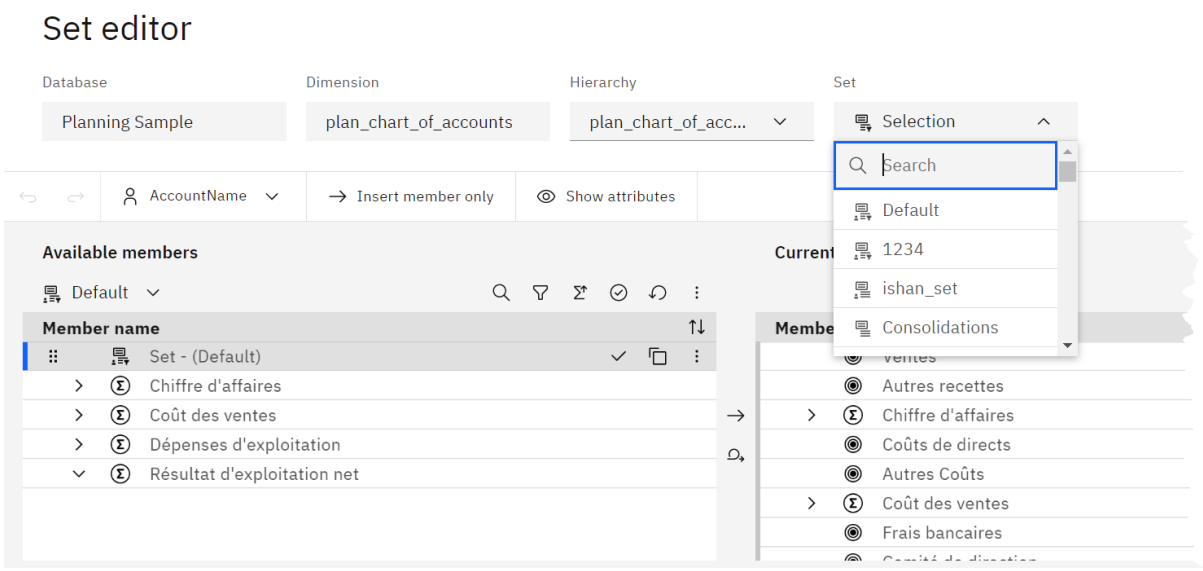
Starting in Planning Analytics Workspace 2.0.80, you can see the Member Unique Name (MUN) for a member when you hover over the member in the **Available members** panel in the set editor.

An MUN is the unique identifier that is used to locate a member in the data source and includes the member name and its ancestry. In the following example, the MUN for **`L Series 1.8 L Wagon 4WD`** displays its dimension (**`model`**), hierarchy (**`model`**), and parent name (**`L Series 4WD`**).




## Search in the Set list on the Set editor

A new search bar in the **Set** list lets you search for a set by name.

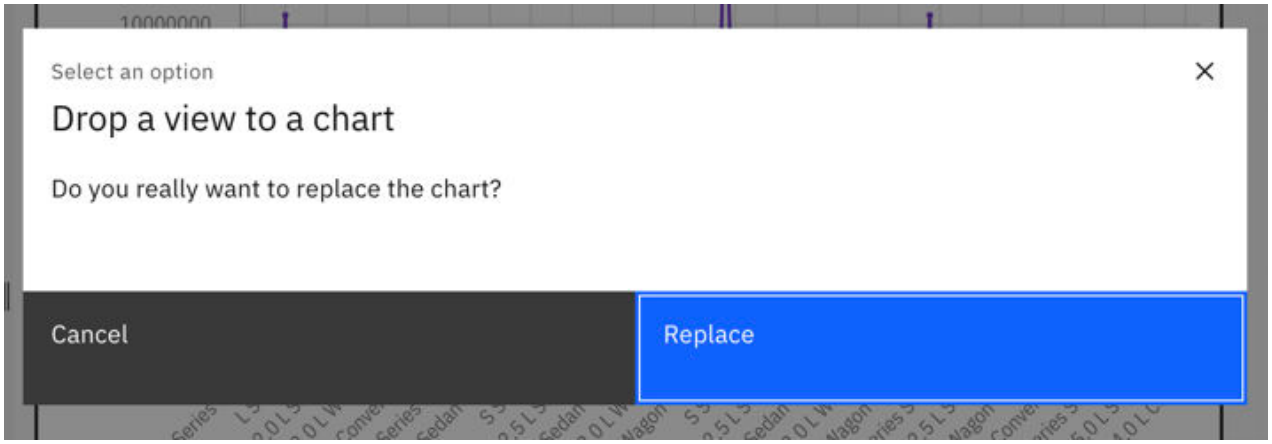



## Drag a view to replace a visualization

You can now drag a cube or a cube view from the data tree onto an existing visualization to replace the visualization. The visualization updates based on the view you drop.

When you drag a view onto an existing visualization, you are asked to confirm if you want to replace the visualization.

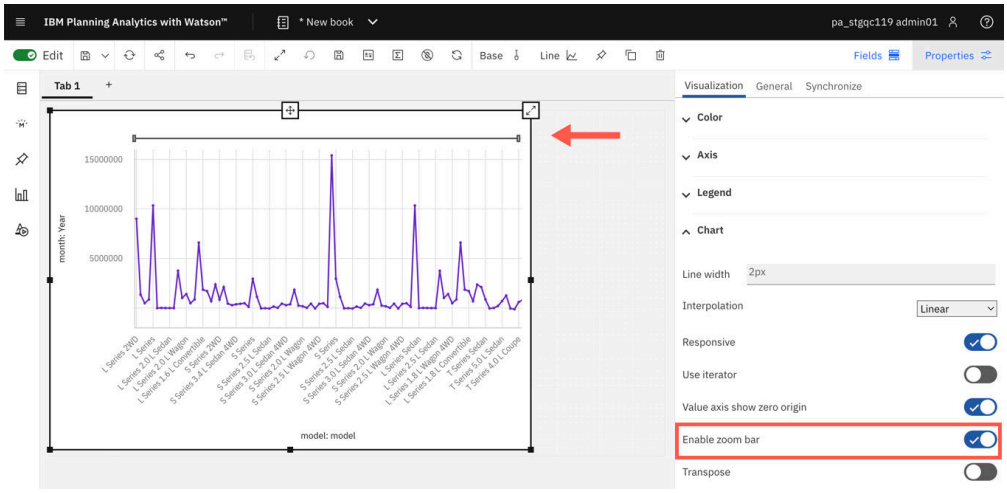
**Note:** Replacing the visualization resets any previous visualization properties and fields that you manually selected or added.




## Zoom bar in Area, Line, and Bar visualizations

A new zoom bar in the **Area**, **Line**, and **Bar** visualization allows you to scroll and focus on a smaller area of the visualization.

Use the **Enable zoom bar** property to add the zoom bar to your visualization. Drag a handle on the bar to zoom and focus on an area in the visualization. The area of focus changes as you drag the handle.




**Note:** When you enable zoom bar, mouse scrolling gets disabled.

## New look for "Save to server" option when saving a view

As part of our continued investment in the IBM Carbon Design methodology, the **Save to server** feature in IBM Planning Analytics Workspace has been updated to use Carbon Design principles.

There is no change to how this feature functions. Views saved to server continue to be available for use in Planning Analytics Workspace, Planning Analytics for Microsoft Excel, and as a process data source.

For more information about Carbon Design principles, see https://www.carbondesignsystem.com.

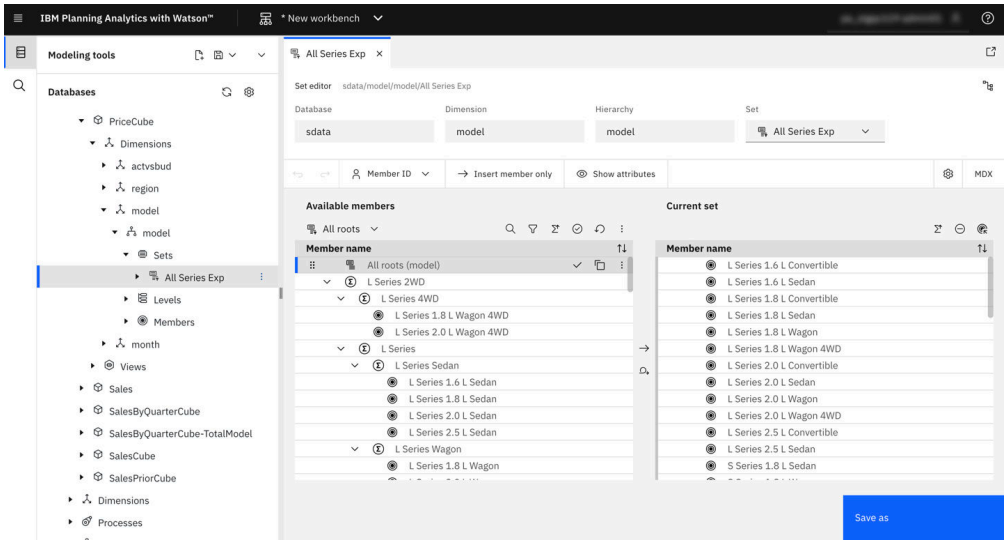
# What's new in modeling - 2.0.80

The following new and changed modeling features are available in Planning Analytics Workspace 2.0.80.

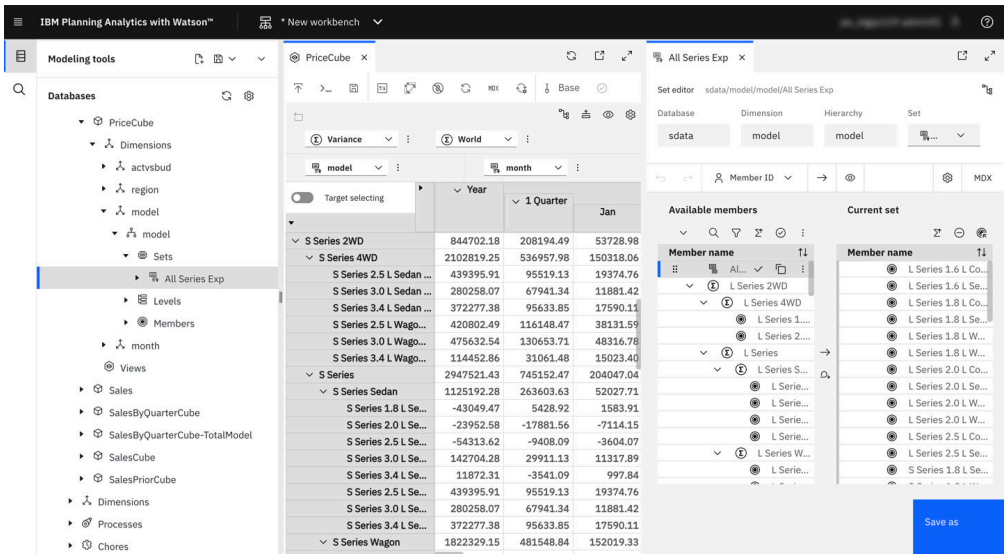
## Support for new experience set editor on the modeling workbench

Set editors that are opened or created from the Databases tree on the modeling workbench now use the new experience interface. Previously, the new set editor was available only through the new experience view. Sets that were opened or created from the Databases tree used the classic interface.

When you open or create a set from the Databases tree, the new experience set editor appears on a new tab on the active tabset, making it easy to compare the set with other assets.




You can open the new set editor in a new window or move the set editor to another region on the workbench to create an editing environment that suits your needs.



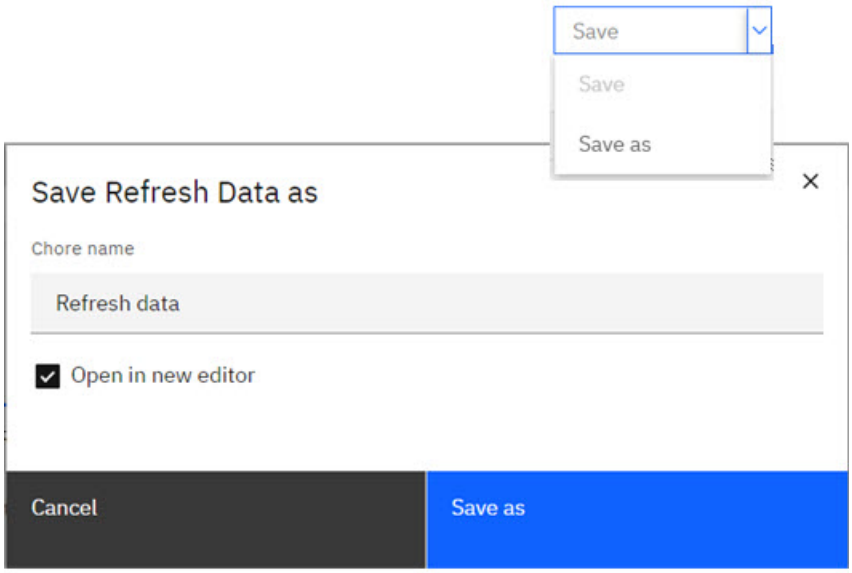

**Note:** When you open a set editor from the new experience view, the set editor opens as a separate dialog and isn't inserted on a new tab in the workbench.

The new experience view and set editor were first made available on the modeling workbench in Planning Analytics Workspace 2.0.78. For more information, see New experience view on modeling workbench.

### Updated chore editor

As part of our continued investment in the IBM Carbon Design methodology and in the interest of providing a consistent user experience across all components, the chore editor in IBM Planning Analytics Workspace has been updated to use Carbon Design principles.

Additionally, the chore editor now includes a **Save as** option that lets you use an existing chore as the basis of a new chore.




### Updated 'Reorder dimensions' option

As part of our continued investment in the IBM Carbon Design methodology and in the interest of providing a consistent user experience across all components, the **Reorder dimensions** page in IBM Planning Analytics Workspace has been updated to use Carbon Design principles.

The **Reorder dimensions** page lets you change the stored order of dimensions in a cube, which can reduce memory consumption and improve performance.

### Check feeders in an exploration on a modeling workbench

If a cube contains a rule with SKIPCHECK and FEEDERS statements, you can use the **Check feeders** option to check that Planning Analytics properly feeds the components of a consolidation.

Right-click a consolidated cell on an exploration, then click **Check feeders** to view a report that indicates if the cell is fed or unfed.

## Deprecation notices - 2.0.80

The following deprecation notices are applicable to Planning Analytics Workspace 2.0.80.

### Deprecation notice - chore editor removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.80

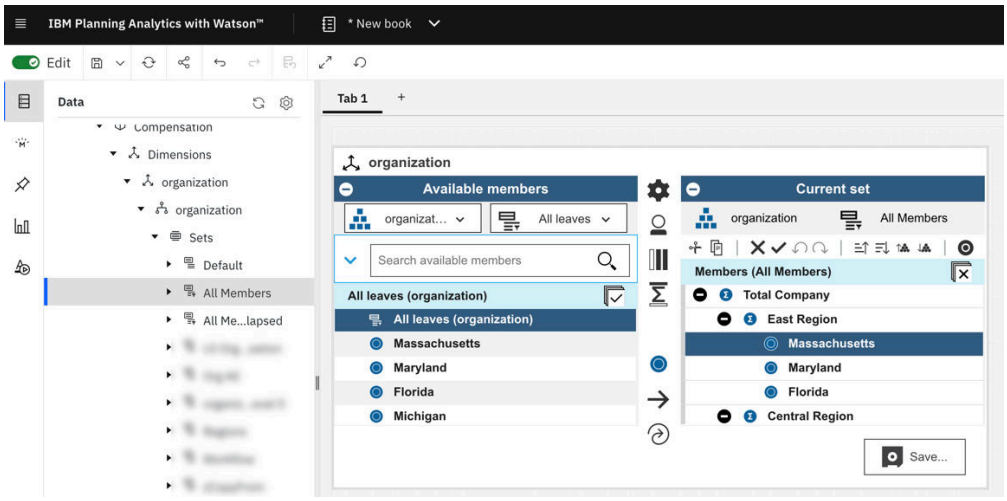
Starting with Planning Analytics Workspace 2.0.80, chores must be created and edited on a modeling workbench.

If you are an administrator or modeler and you attempt to open a book containing a chore in 2.0.80, you receive a notification that chore editing must be performed on a workbench. You can click **Open in workbench** to immediately open the chore on a new workbench.

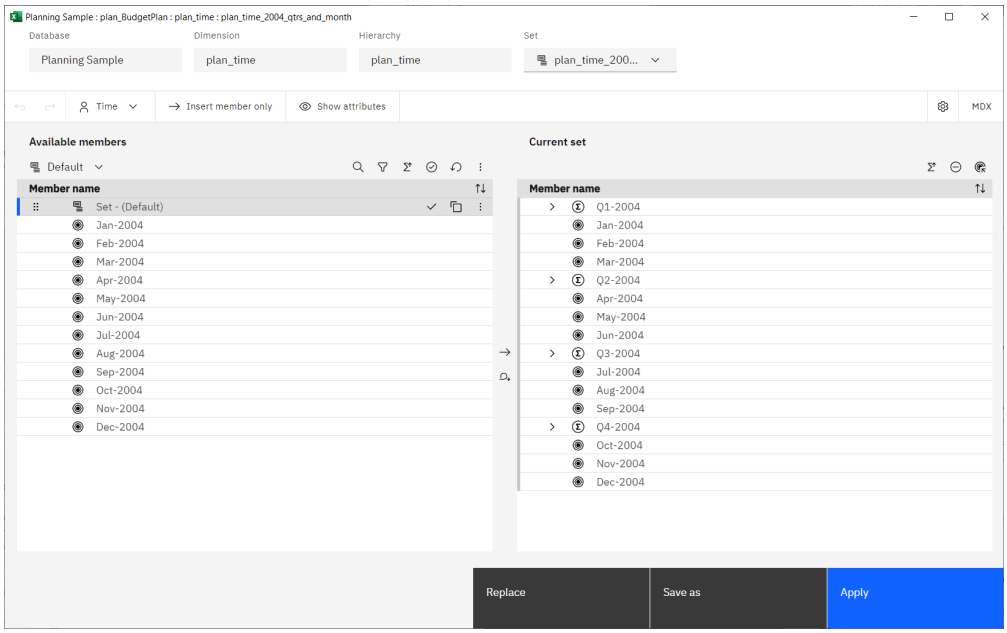
## Deprecation notice - set editor in a widget to be removed from books as of Planning Analytics Workspace 2.0.81

The ability to add a set editor in a widget from the data tree in Books will be deprecated in Planning Analytics Workspace 2.0.81. Currently, when you select **Edit set** from the data tree, the classic experience set editor opens in a widget.

**Classic set editor in a widget:**




Starting in Planning Analytics Workspace 2.0.81, when you edit a set from the data tree, the new experience set editor will display. The new set editor opens as a dialog and is not inserted as a widget in the Book.




The new experience set editor was introduced in Planning Analytics Workspace 2.0.71. For more information, see New experience view and set editor.

# 2.0.79 - What's new, August 25, 2022

Learn about new features and known issues in version 2.0.79 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

## Planning Analytics Workspace Classic on Cloud no longer available as of 2.0.79

The 'new experience' is a major update to the Planning Analytics Workspace user experience that was originally released as part of the 2.0.56 update in October 2020.

ThePlanning Analytics Workspace new experience provides an updated look and feel, improved navigation, and better content management. It also introduces Applications and Plans, a dedicated modeling environment, and predictive forecasting capabilities.

The user experience in 2.0.55 and earlier releases is referred to as Planning Analytics Workspace Classic. Cloud environments provisioned before the 2.0.56 update could remain on Planning Analytics Workspace Classic, preview the new experience, or permanently switch to the new experience.

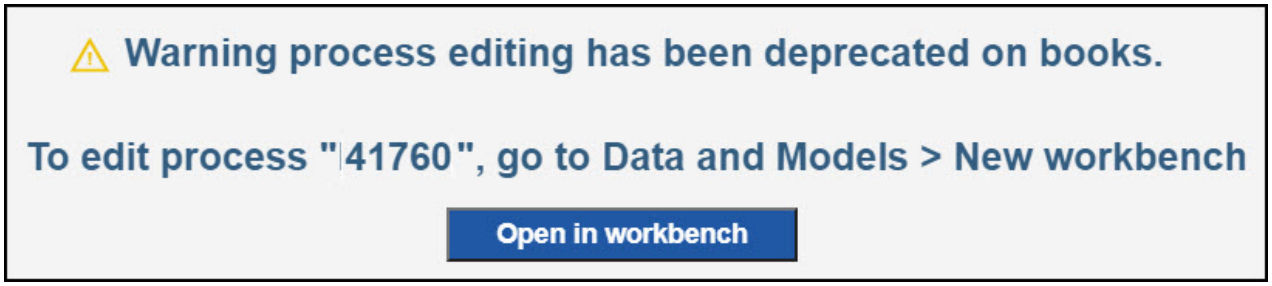
As previously and repeatedly announced in the monthly Planning Analytics Workspace Early Notification and in the monthly What's New topic on IBM Documentation, Planning Analytics Workspace on Cloud environments that use the Classic interface will automatically be updated to the new experience as part of the 2.0.79 update.

This announcement does not impact Planning Analytics on-premises deployments.

## Process editor available only on modeling workbenches

As previously announced, processes must be created and edited on a modeling workbench as of Planning Analytics Workspace 2.0.79.

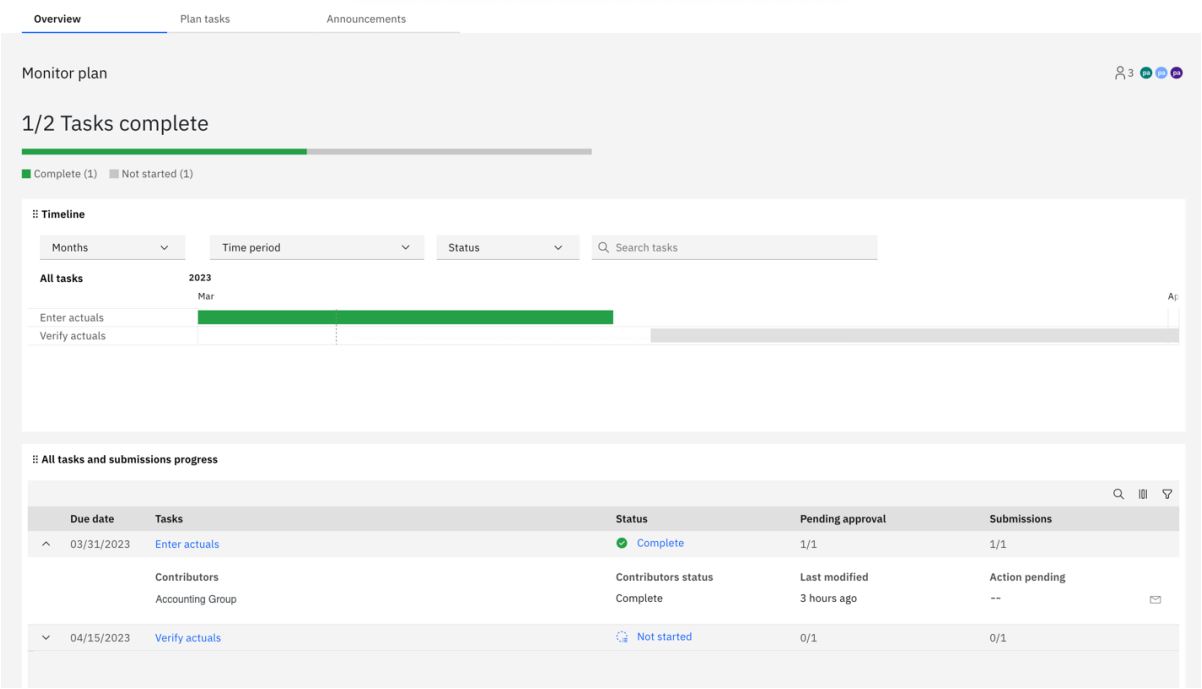
If you are an administrator or modeler and you attempt to open a process editor that was saved in a book prior to 2.0.79, you receive a notification that process editing must be performed on a workbench. You can click **Open in workbench** to immediately open the process on a new workbench.




## Track and report on plan progress with the plan Overview report

You can view a plan-level overview report to help you assess the progress of your plan.

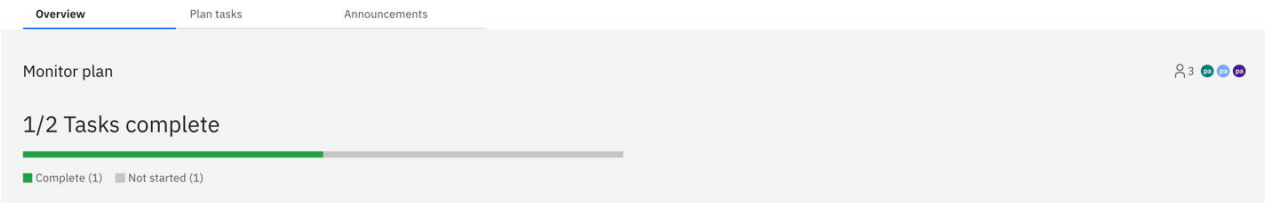
The report appears on the **Overview** tab when you are viewing plan details.




The overview report consists of a progress bar and two panels: **Timeline** and **All tasks and submissions progress**. You can click the handle next to the panel name to drag a panel to a different location on the Overview report.

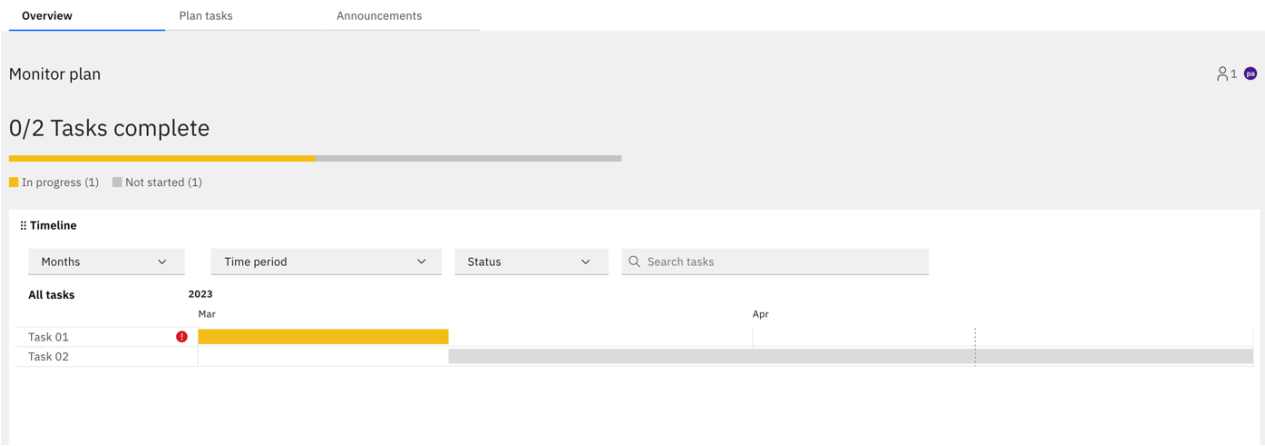
## Progress bar

The progress bar displays the progress of plan. You can see the number of tasks that were completed and the number of tasks that have not yet started.




## Timeline panel

The **Timeline** panel shows the status of tasks within a plan against a timeline, which can be configured to show either months or weeks.



If a task has only a start date or only an end date defined, the date is represented by a diamond on the timeline. Hover the pointer over the diamond to view the exact start or end date.

If a task has both a start date and an end date defined, a bar represents the date range on the timeline. Hover the pointer over the bar to view the start and end date of the task.

Any task that is not complete and overdue is identified by an urgent notice icon .

The **Timeline** panel includes advanced filtering. You can filter tasks by name, task status, and time period.

### All tasks and submissions progress panel

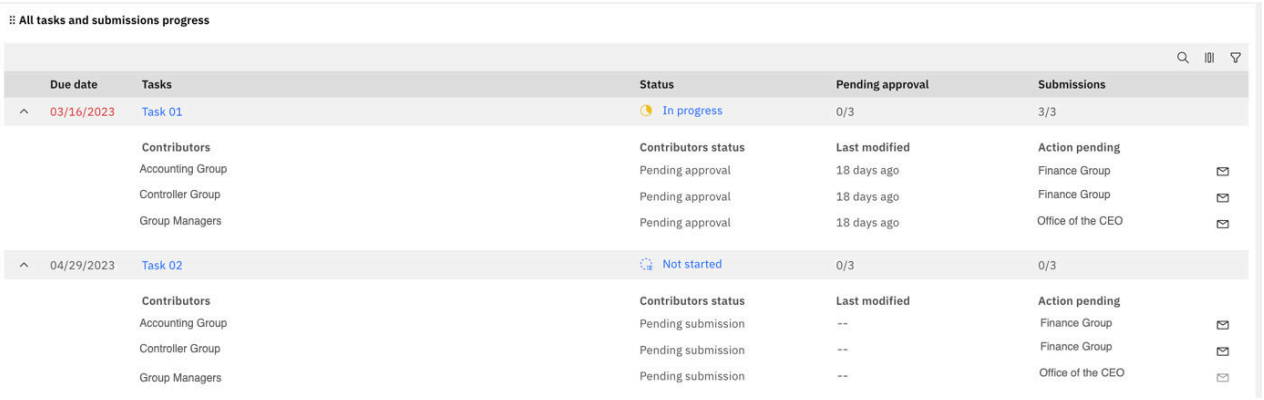
The **All tasks and submissions progress** panel displays progress for all submissions and individual tasks in a plan.

Expand each task to see status details for the individual submissions in that task.

Hover the pointer over a task name in the **Tasks** column to view the task description.

You can sort on any table heading and also search for tasks by name. You can customize the **All tasks and submissions progress** panel by choosing the columns that you want to see and filtering by task status.

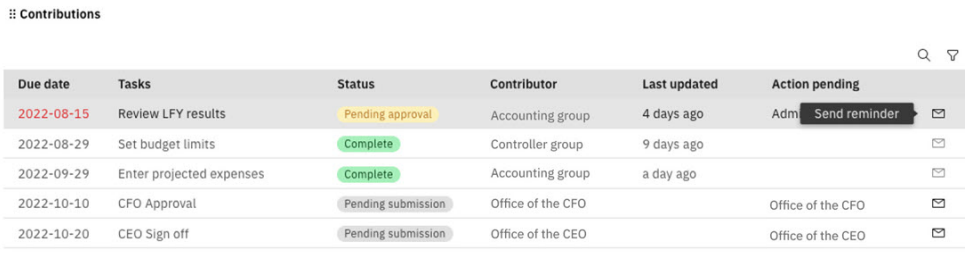
When an individual submission is in the **Pending submission** or **Pending approval** state, an email icon is active at the end of the submission record. Click the email icon to send a reminder that action is required to members of the group that is identified in the **Action pending** column.




## Send email reminders to contributors of open tasks

You can now send an email reminder to plan contributors from the **Contributions** panel of the **Overview report**.

When an individual contribution on the **Contributions** panel is in the **Pending submission** or **Pending approval** state, an email icon is active at the end of the contribution record. Click the email icon to send a reminder that action is required to members of the group identified in the **Action pending** column.

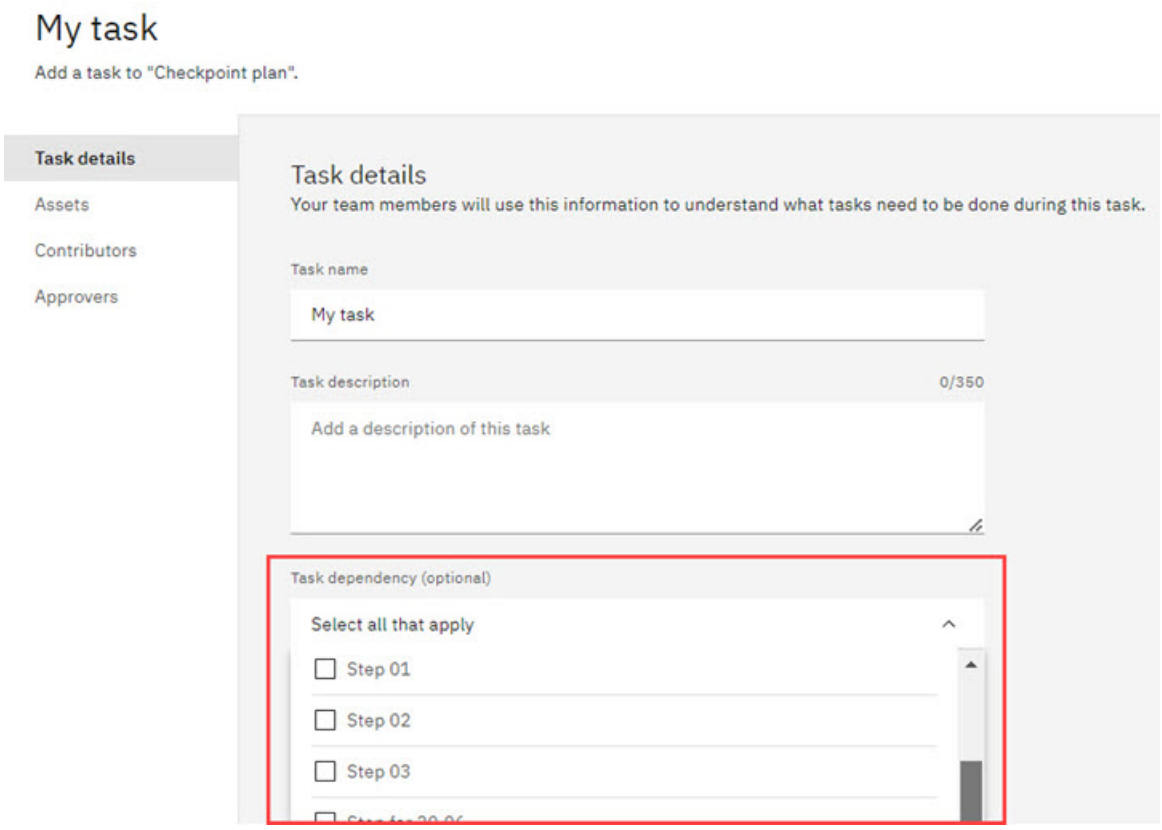


## Set task dependencies for plan automation

You can now automatically open a task upon completion of one or more dependent tasks.

When you create or edit a task, use the **Task dependency** option on the **Task details** tab to set the dependencies.

You can select one or multiple tasks as dependencies. If you select a single dependency, the task is automatically opened when the single dependent task is complete. If you select multiple dependencies, the task is automatically opened only when all dependent tasks are complete.



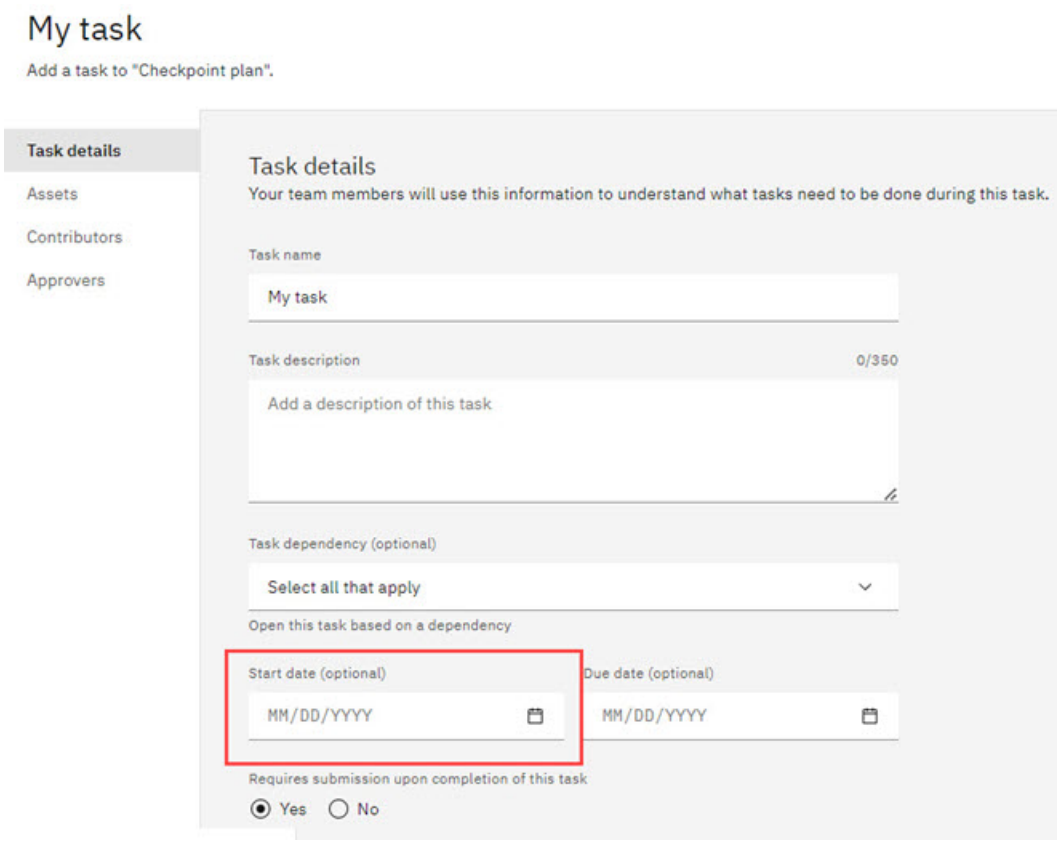

When a task is configured to use dependencies, the dependencies are identified on the task details view




## Set a start date for your tasks

You can now set a start date as well as an end date for a task.

Previously, only an end date could be set. Set the start date on the **Task details** tab when creating or editing a task.




## Use the new Features and Configuration tile to administer your Planning Analytics Workspace on Cloud environment

The **Features** tile in Planning Analytics Administration on Cloud is renamed as **Features and Configuration**. When you click the **Features and Configuration** tile, you see two tabs: **Features** and **Configuration**.

Use the **Features** tab to manage how your users are exposed to new features that are subject to enablement by a Planning Analytics Workspace administrator, as described in Manage features.

Use the **Configuration** tab to configure various aspects of the Planning Analytics on Cloud environment. For example, you can use the **Configuration** tab to set an inactivity timeout for Planning Analytics for Excel and Planning Analytics Workspace users in your environment.

**Note:** The **Features** tile name and capabilities remain unchanged in Planning Analytics Administration Local, but the interface has been updated to provide a user experience that is consistent with the rest of Planning Analytics Workspace.

## Configure an inactivity timeout for Planning Analytics on Cloud

Planning Analytics Workspace on Cloud enforces an inactivity timeout. Previously, this inactivity period was set to 60 minutes for all users and could not be configured by a Planning Analytics administrator.

### About this task

As of Planning Analytics Workspace 2.0.79, the inactivity timeout is configurable. When a user is inactive for the period of time specified by the Planning Analytics Workspace administrator, the user is disconnected from the Planning Analytics database. This inactivity timeout is mandatory and is applicable to all Planning Analytics Workspace on Cloud users, as well as Planning Analytics for Excel users.

For details on configuring timeouts in Planning Analytics Workspace Local, see Configuring parameters for Planning Analytics Workspace Local.

To set an inactivity timeout:

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace home page.
2. Click the **Features and Configuration** tile.
3. Click the **Configuration** tab.
4. Click the **Inactivity timeout** configuration parameter.
5. Use the **Hours** and **Minutes** controls to set an inactivity timeout period. The minimum timeout period is 30 minutes and the maximum period is 24 hours.

## Use the new interface to manage chores involved with a specific process

Previously, when you opened the context (right-click) menu for a process in the Databases tree, you could click the Display chores involved option to open the **Modify chores** window. This window listed all of the chores that included the process. You could then use the **Modify chores** window to edit selected chores that included the process or delete the process from selected chores.

### About this task

Planning Analytics Workspace 2.0.79 simplifies chore management by presenting two distinct options when you open the context menu for a process in the Databases tree: **Edit related chores** and **Remove from chores**.

## Edit related chores that involve a specific process

### Procedure

1. Right-click the process on the **Databases** tree in a modeling workbench.
2. Click **Edit related chores**. If the process is not included in any chores, you receive notification and can cancel the procedure. Otherwise, continue to the next step.
3. On the **Edit chores** window, select the chores you want to edit.
4. Click **Edit**.

   Each of the chores you selected opens for editing on a separate tab on the modeling workbench.

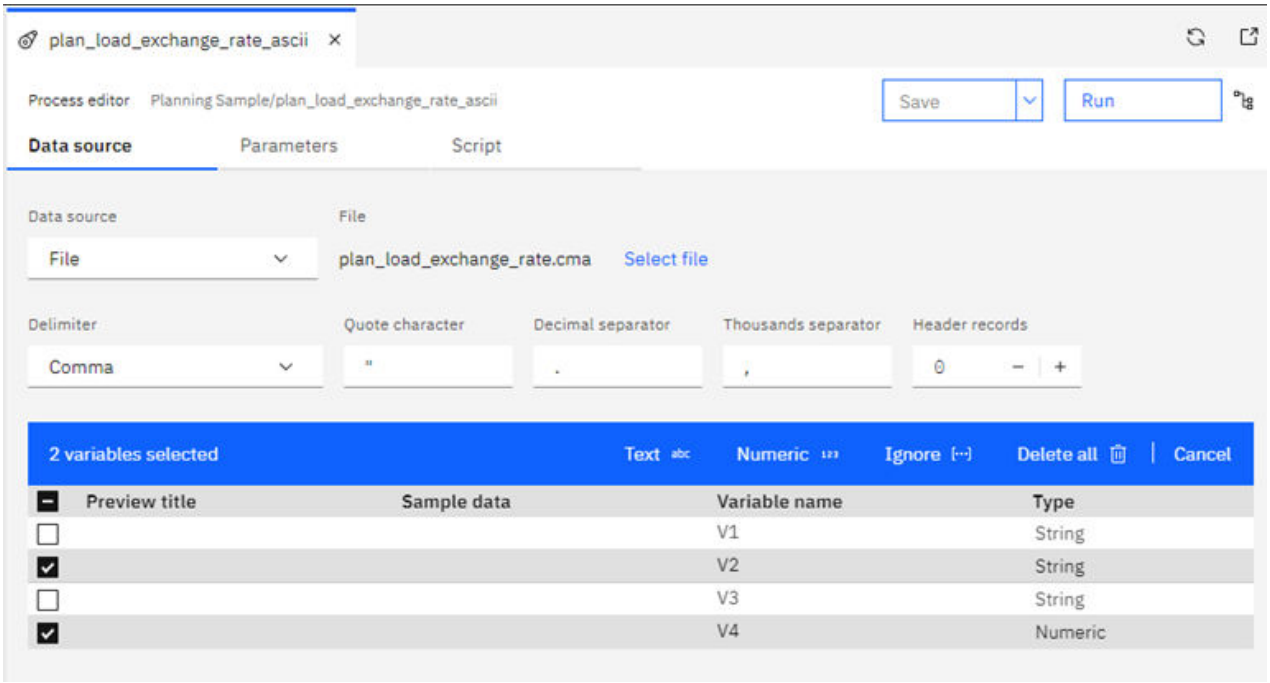
## Remove a specific process from selected chores

### Procedure

1. Right-click the process on the **Databases** tree in a modeling workbench.
2. Click **Remove from chores**. If the process is not included in any chores, you receive notification and can cancel the procedure. Otherwise, continue to the next step.
3. On the **Remove chores** window, select the chores from which you want to remove the process.
4. Click **Remove**.
5. Click **Delete** when prompted for confirmation of removal.

## Select multiple variables in the process editor

You can now select multiple variables in the process editor. When multiple variables are selected, you can apply any option on the selected variables bar to multiple variables simultaneously.

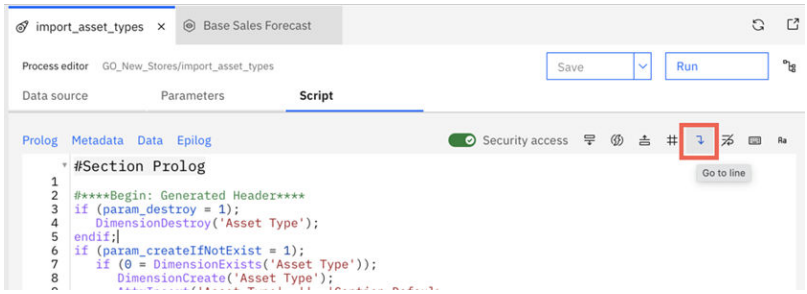



## Go to a specific line in the process editor

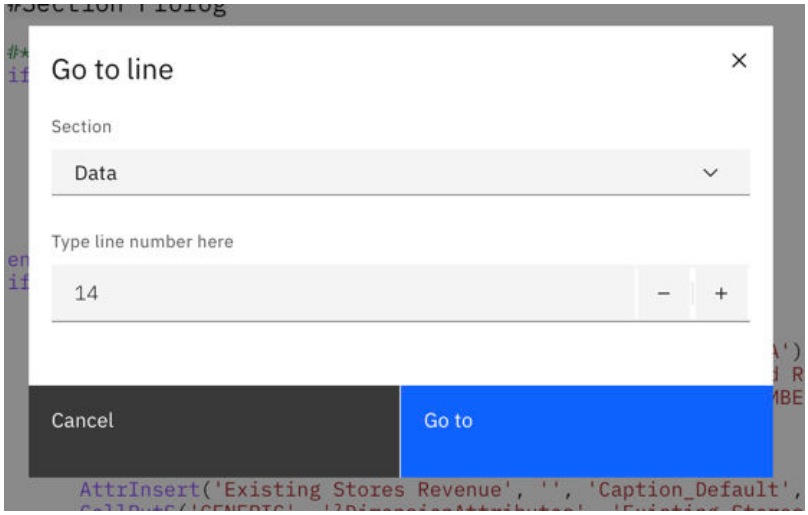
You can now jump to a specific line in the process editor with **Go to line**. This capability was previously available only in the rules editor through shortcut keys (Ctrl + L or Cmd + L). Now, you can click the **Go to line** icon in both the rules and process editors to quickly move up and down the script. This feature is especially useful if you encounter a process error that requires you to go to a specific section and line number.

To use the **Go to line** feature in the process editor:

1. On the **Script** tab, click the **Go to line** icon.




2. Select the section and enter the line number that you want to go to.




**Note:** If you enter a line number that does not exist in the section, an error displays which shows you how many lines are in that section.

3. Click **Go to**. The cursor moves to the start of that line.

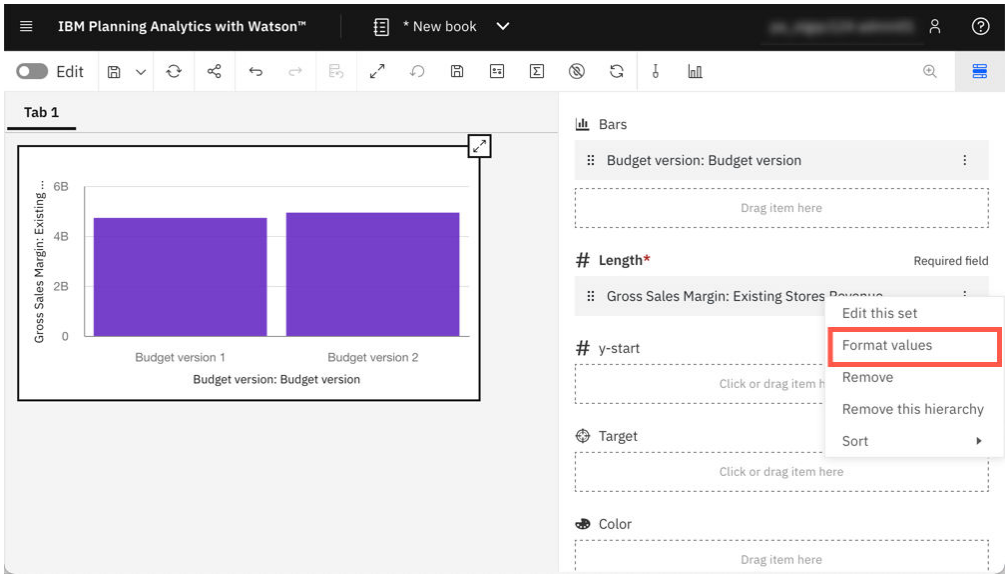
Note, the **Go to line** dialog in the process editor is slightly different from the one in the rules editor. Since the script in the process editor has sections, you need to specify both the section and the line number you want to go to. Whereas in the rules editor, you need to enter only the line number you want to go to.

## Apply formatting to visualization axis

When you apply formatting to a visualization axis now, the axis retains the format when you change the member on the axis or when synchronization automatically changes the member. The formats are also retained when you switch from one visualization to another or convert the visualization to an exploration and vice versa.

To apply formatting to a visualization axis:

1. Select the visualization and click **Fields**.
2. Click the menu icon for the axis that you want to format and select **Format values**.




3. Choose the format that you want to apply to the axis.

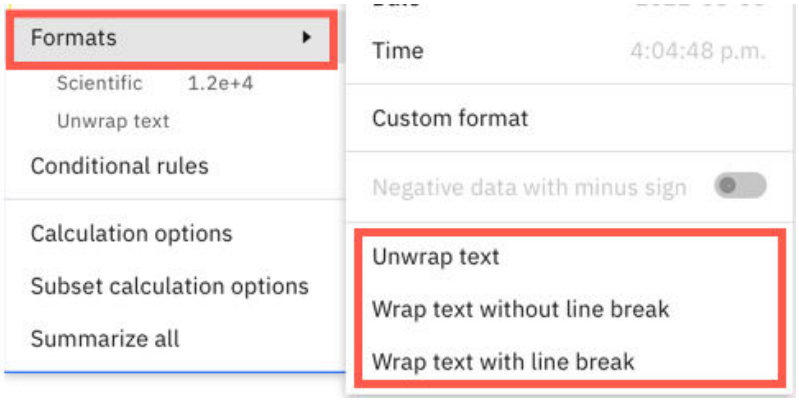
You can also right-click the axis title in a visualization and click **Format values** to select and apply a format to the axis.

## Wrap text in data cells

You can now format string cells in explorations (or views) to wrap text. You can wrap multi-line text in data cells with or without line breaks. Previously, you could only wrap text in headers.

**Note:** The wrap text feature is only supported for text that is entered in a cell using the text editor (press F2 or click the pencil icon in the cell).

To wrap text, right-click a row or column. Go to **Formats** and select how you want to wrap text for that row or column.




To unwrap text, right-click the row or column and select **Unwrap text** under **Formats** or go to **Clear** and click **Formats**.

## Deprecation notice - chore editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.80

Starting with Planning Analytics Workspace 2.0.80, chores must be created and edited on a modeling workbench.

If you are an administrator or modeler and you attempt to open a book containing a chore in 2.0.80, you'll receive a notification that chore editing must be performed on a workbench. You can click **Open in workbench** to immediately open the chore on a new workbench.

# 2.0.78 - What's new, July 13, 2022

Learn about new features and known issues in version 2.0.78 of IBM Planning Analytics Workspace in the following topics.

**Important:** The July 13, 2022 update includes only Planning Analytics Workspace Local 2.0.78. The Planning Analytics Workspace Cloud 2.0.78 update is targeted for July 19, 2022.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

### Timeline of previous Planning Analytics releases

For a comprehensive listing of Planning Analytics component releases by month and year, see Timeline for IBM Planning Analytics Releases. You can view versions and release months for the following components:

- Planning Analytics Local TM1 Server

- Planning Analytics Workspace
- Planning Analytics for Microsoft Excel
- Planning Analytics Spreadsheet Services/TM1 Web

# Rules must now be created and maintained on a modeling workbench

As of Planning Analytics Workspace 2.0.78, rules must be created and edited on a modeling workbench.

If you are an administrator or modeler and you attempt to open an existing rule in a book in 2.0.78 or later, you receive a notification that rule editing must be performed on a workbench. You can click **Open in workbench** to immediately open the rule on a new workbench.

All options to open or edit rules have been removed from the Databases tree in books.

# New experience view now on modeling workbench

The new experience view (or exploration) and new set editor are now available on the modeling workbench. Along with an improved look and feel, this new experience interface has several usability improvements and enhancements. Until now, the new experience interface was only available in Planning Analytics Workspace books.

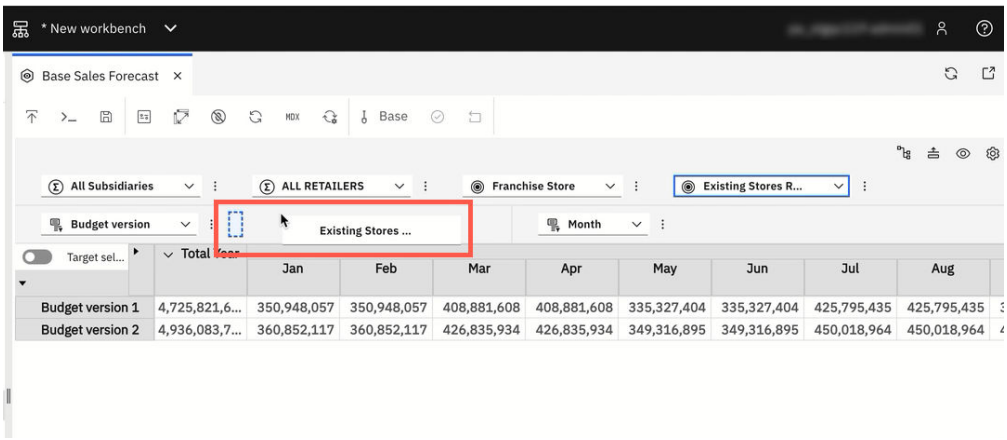
**Important:** All existing and new views on the workbench now use the new experience interface.

## Using the new experience view

The new experience view offers flexible options to view, analyze, and organize data. Features in the new experience view are also now more accessible.

- **A more guided and intuitive experience**

  The new view's improved design offers a more guided experience without compromising its capabilities. For example, the new view now guides you to the drop zone when you drag a dimension to a different location in the grid.



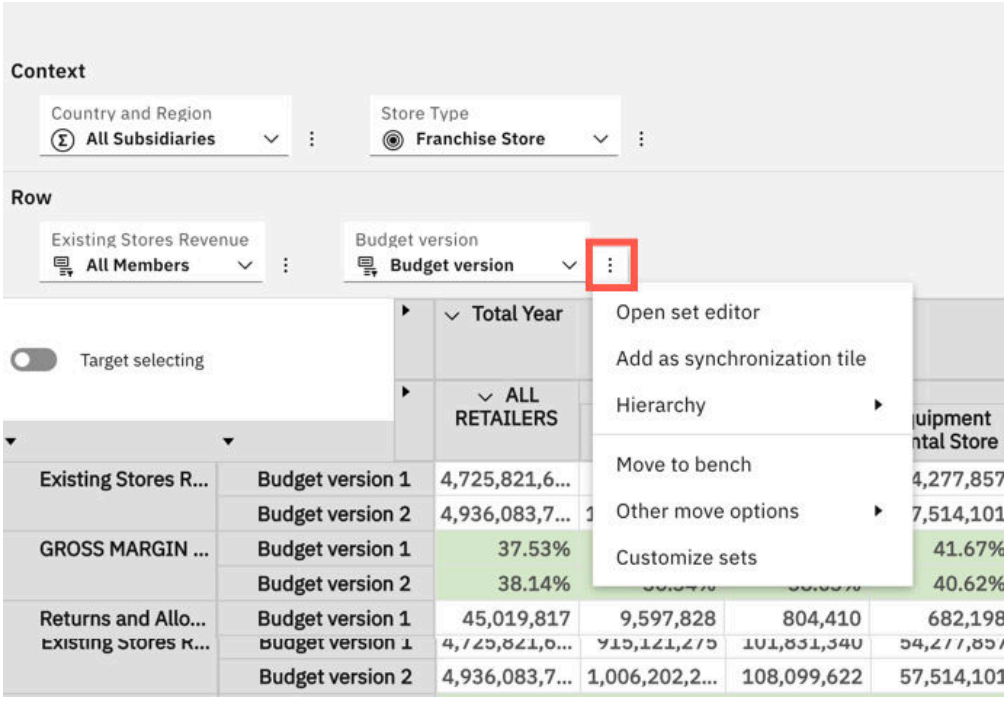

You can use the icons in the grid  to:

  - See the cube in the data tree.
  - Expand or collapse the overview.
  - See benched dimensions.
  - Select hierarchy, iconography, context information, and section headers to customize your view.

  Each dimension tile in the grid now has a menu icon next to it from which you can:

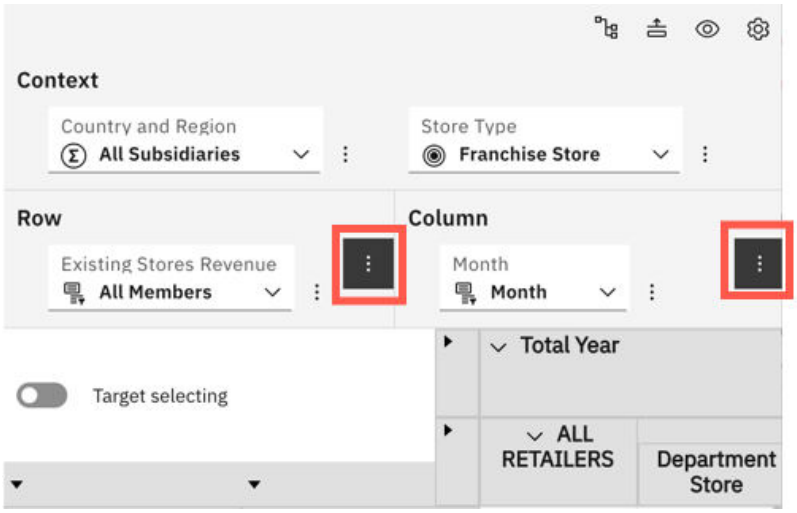
  - Open the set editor.

- Remove, replace, or add related hierarchy.
- Move the dimension to the bench.
- Manually move the dimension right or left, or to a row, column, or context.
- Limit the number of sets for a dimension (**Customize sets**)



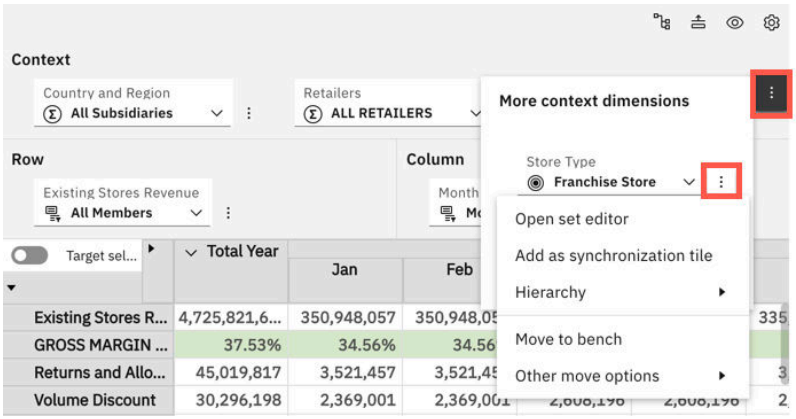

Depending on the number of dimensions or the size of the view, all dimensions might not display.

To see dimensions that are hidden from the view, click the **More dimensions** menu in the **Context**, **Row**, or **Column** area.



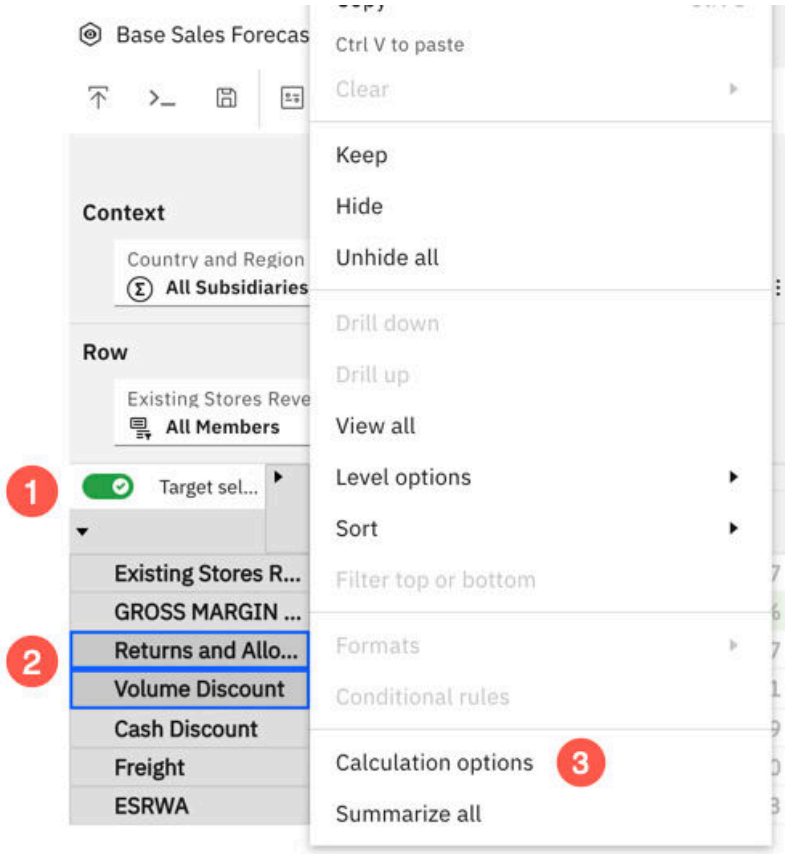

From the **More dimensions** menu, you can drag dimensions to the **Context**, **Row**, or **Column** area and vice versa. You can also move dimensions that appear within the **More dimensions** menu.

You can click the menu next to the member's name to open the set editor, move the dimension to the bench, see move options, and more.




- **Target selecting toggle for asymmetric calculation**

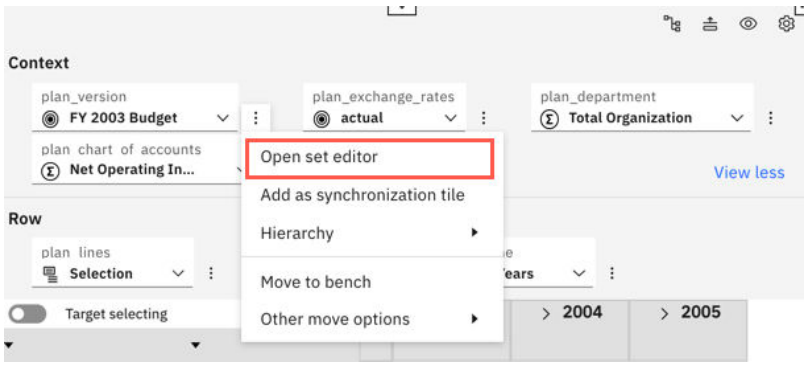
  The new **Target selecting** toggle is used to create an asymmetric calculation. You no longer need to select the individual button on the row or column headers to create an asymmetric calculation. Instead, enable **Target selecting** and select the members that you want to use in the asymmetric calculation. Right-click a selected member and click **Calculation options**.



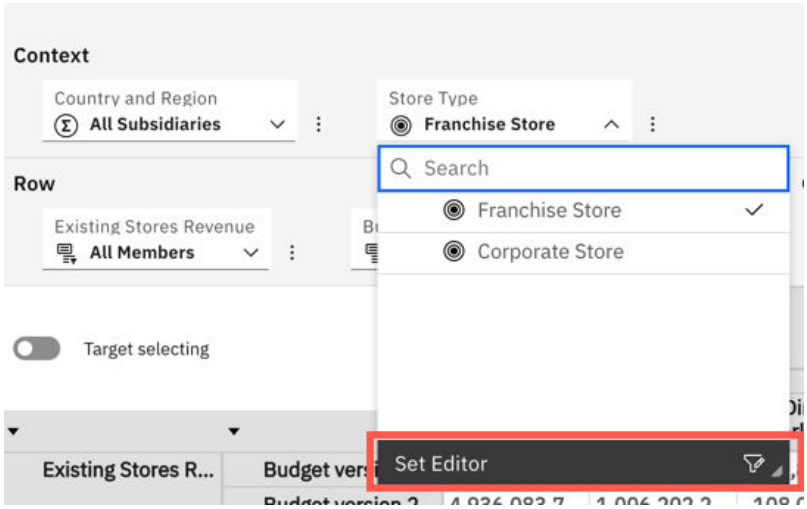

## Using the new set editor

The new set editor can be accessed only through the new view. If you open a set as a separate tab from the data tree, it still uses the classic interface.

To open the set editor, click the menu icon next to a dimension in the view and select **Open set editor**.



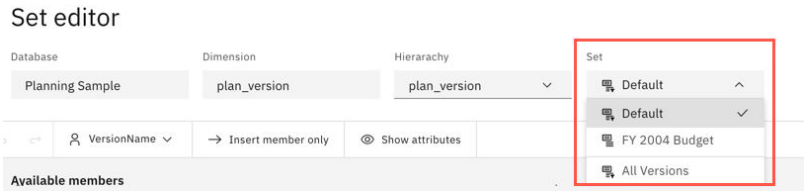

You can also double click the dimension tiles or click the **Set Editor** icon in the dimension menu to open the set editor.




As with the new view, the new set editor offers an improved flow, a more intuitive experience, and flexible options to edit your set.

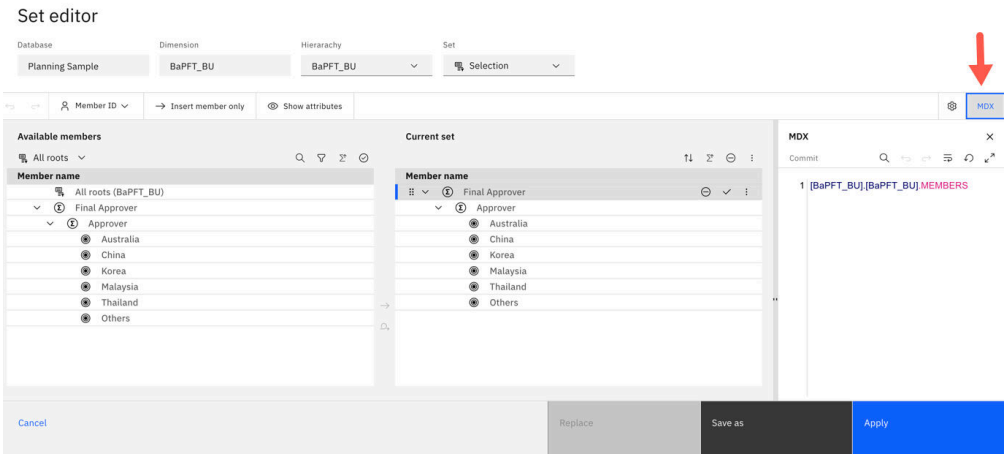
- **Set menu**

  With the **Set** menu, you can now change the set from within the set editor.




- **Improved MDX panel**

  Click **MDX** to see the **MDX** panel and script. You can edit the script in the panel and click **Commit** to apply your changes to the current set.

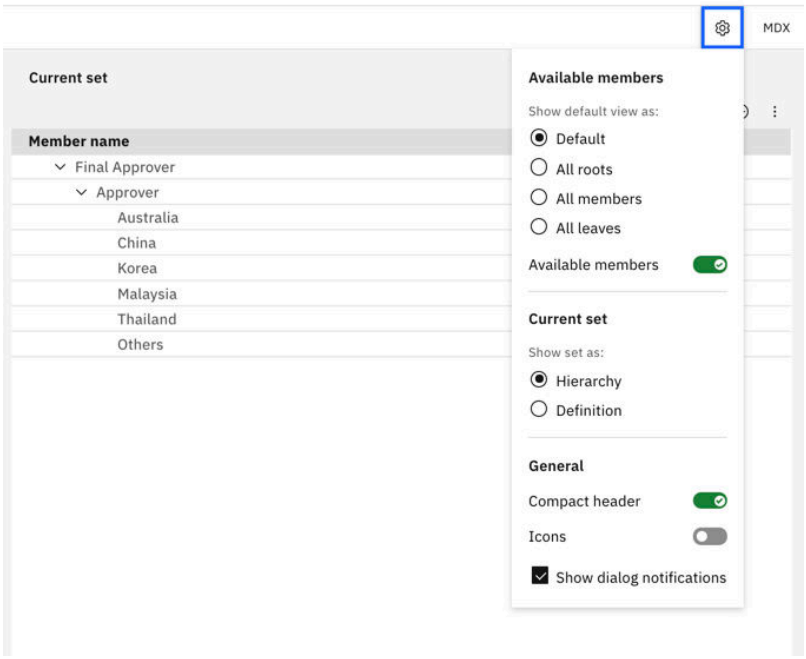



- **Settings menu**

  The **Settings** menu offers options to customize the display for your set.

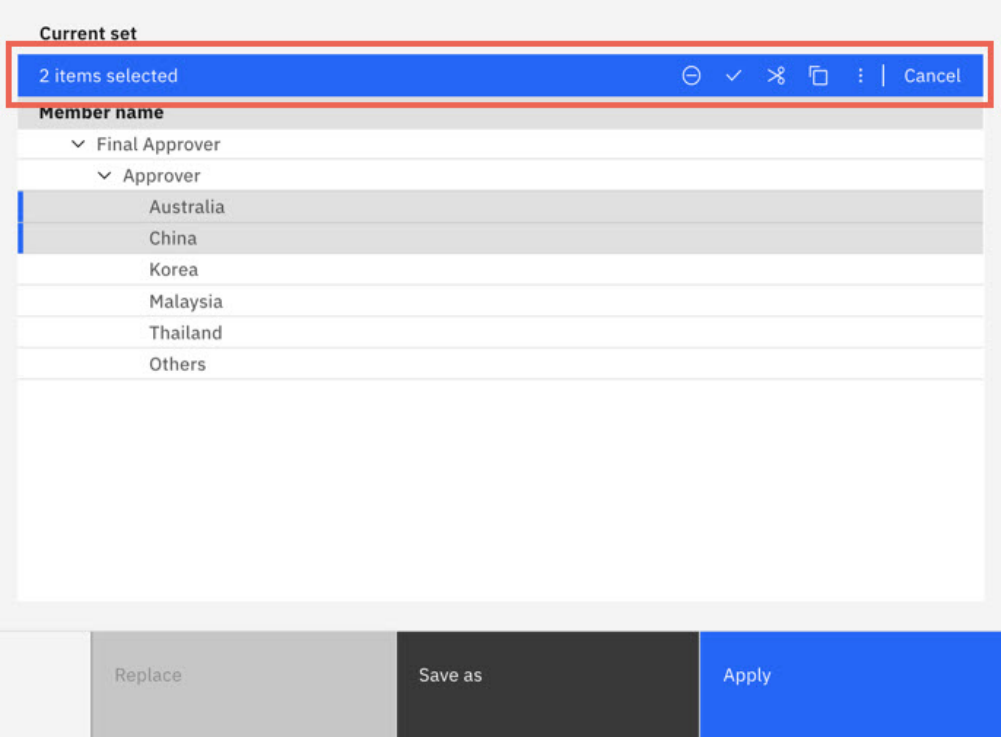
  Enable **Compact header** under the **General** section to reduce the size of the set editor header and see more members.

  You can also remove icons from all members for a simplified display with the **Icons** property under **General**.



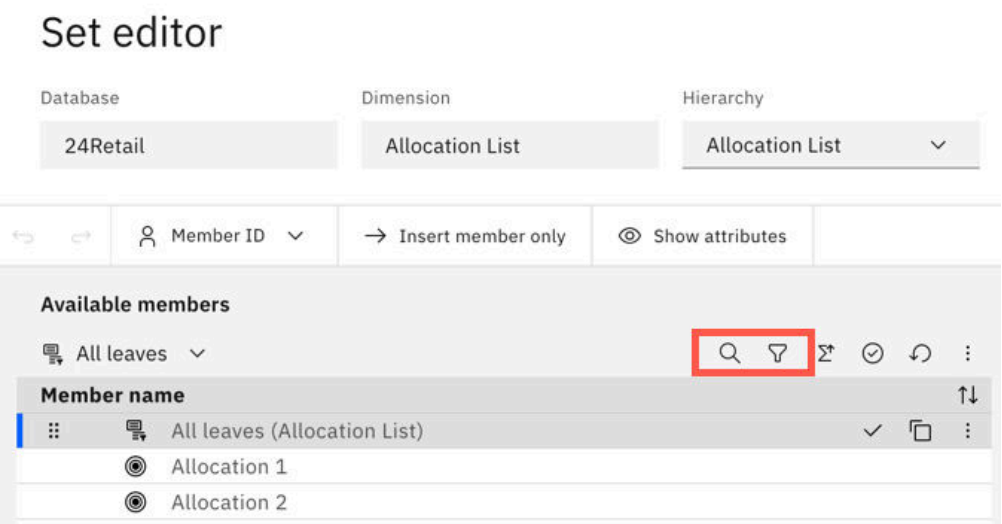

- **Multi-select activity display**

  When you select multiple members in **Available members** or **Current set**, an activity bar now displays with your selection information and available options.




- **Improved features**

  The Search and Filter features in the new set editor are now more intuitive and improved for usability. The filter feature allows you to filter a set by one or more attributes to choose the available members that appear in the set. The search feature lets you search for members in the set editor. You can use both features to include members in a set that match specific criteria.




The new experience set editor also offers the ability to delete sets from within the set editor.

## Approvals and Rejections in Plans

Approvals and rejections are now available in the plans submission process. In addition, improvements to email notifications and the overall submission process make it easier to work with plans.

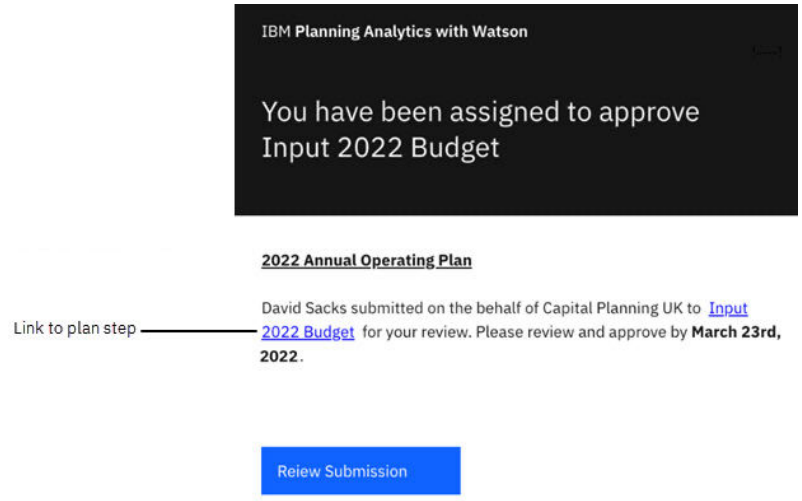
When you create a plan, you can now add approvers for each contribution group. The assigned approvers have the authority to accept or reject submissions. This is an improvement over previous versions of Planning Analytics Workspace, where an owner or co-owner could only reset an unsatisfactory submission.

Event-triggered email is now sent to relevant plan users when any of the following events occur in a plan. The email includes details on the event and, when applicable, a link to the location in the plan where you can take further action or review more information.

*Table 1. Event-triggered email recipients*

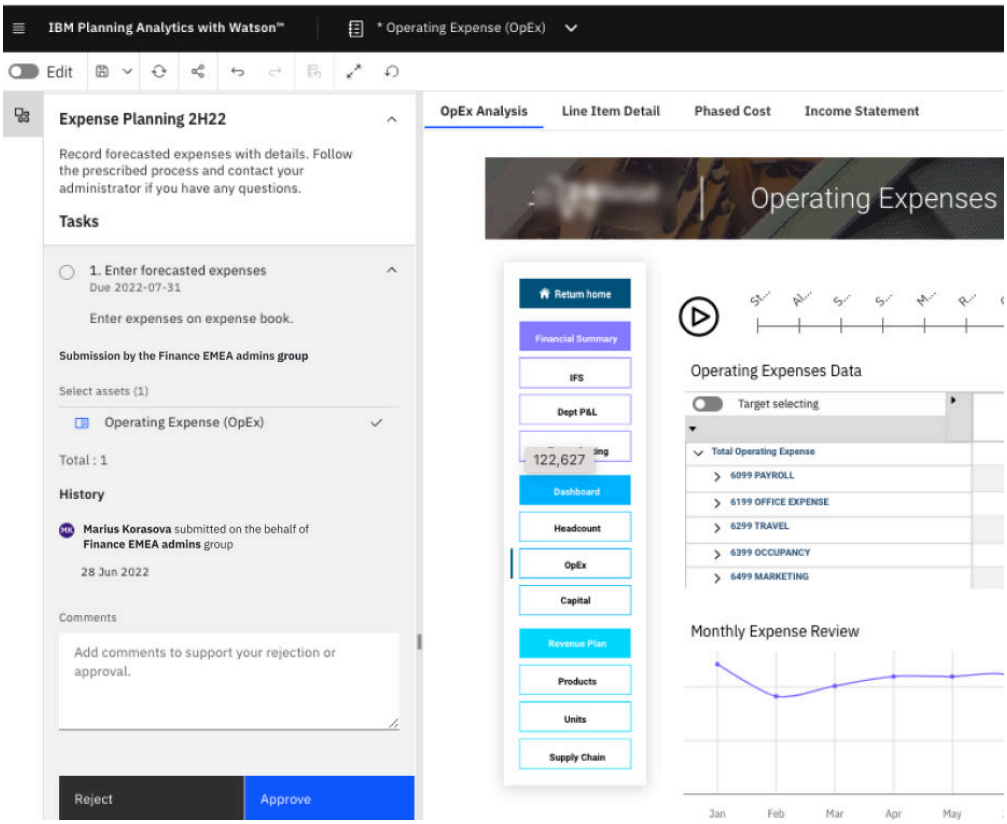
| **Event** | **Email recipients** |
|---|---|
| A plan step is opened | Members of all groups that are assigned to the step |
| A plan step is submitted | Approvers that are assigned to the group that has submitted the step<br>The plan owner and co-owners |
| A submitted step is rejected | The submitter of the rejected step |
| A submitted step is approved | The next in line approver, if any<br>The plan owner and co-owners |

For example, when a plan step is submitted, approvers who have been assigned to the group that submitted the step receive an email similar to this. The email would indicate the actual *step name*, *step contributor name*, and *group name* for the submission. The approver can click the **Review submission** button in the email to open the submission and either approve or reject it.



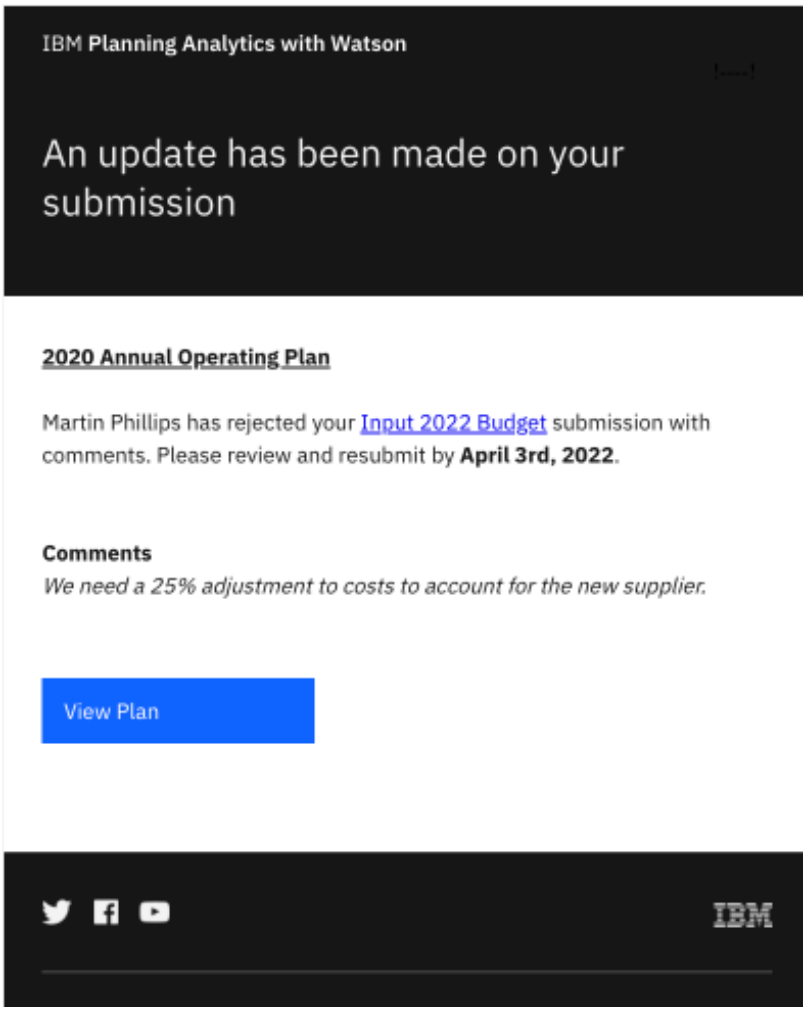

In addition, the submission process has been enhanced to provide more insight and options for owners, contributors, and approvers.

- You can now assign one or more approvers to a plan step. Approvers review step submissions and determine the disposition of the submissions.

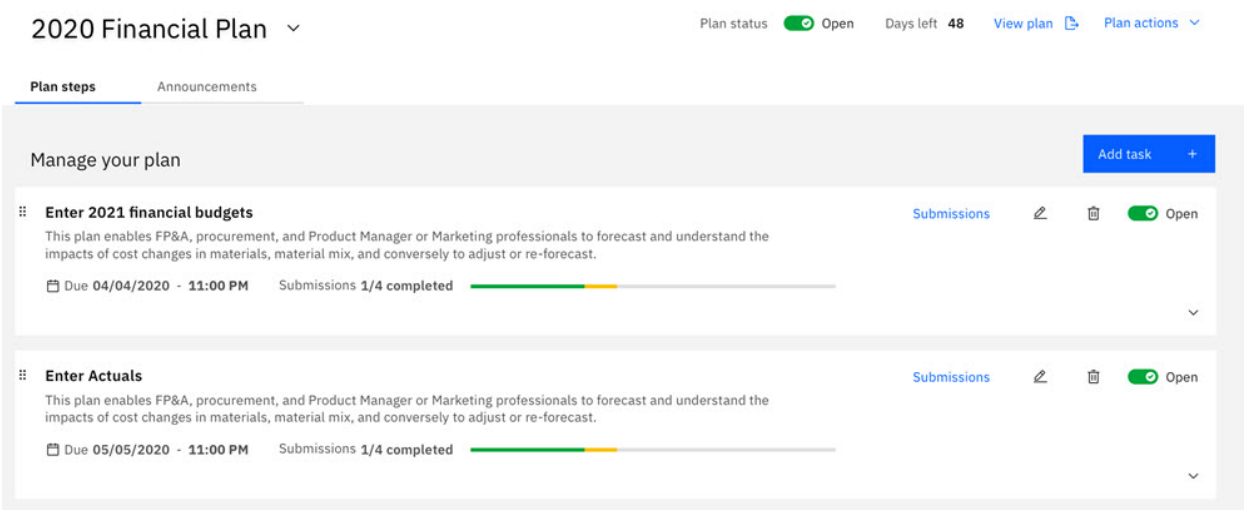



- When you add an approver to a step, the approver name is prefixed with a number in the **Approver** list. These numbers represent an approval hierarchy. When approver 1 approves a submission, it is routed to approver 2, and so on until the last approver approves the submission.
- If a submission is approved, the step remains closed for submissions to the group who submitted it.

  If a submission is rejected, the step is re-opened for the group who initially committed the submission. However, the assets included in the step are not reset; contributors can resume working on the assets in the same state they were in when the initial submission was made.




- The plan page shows the state of submissions for each step. A submission is not identified as completed until it has been approved.

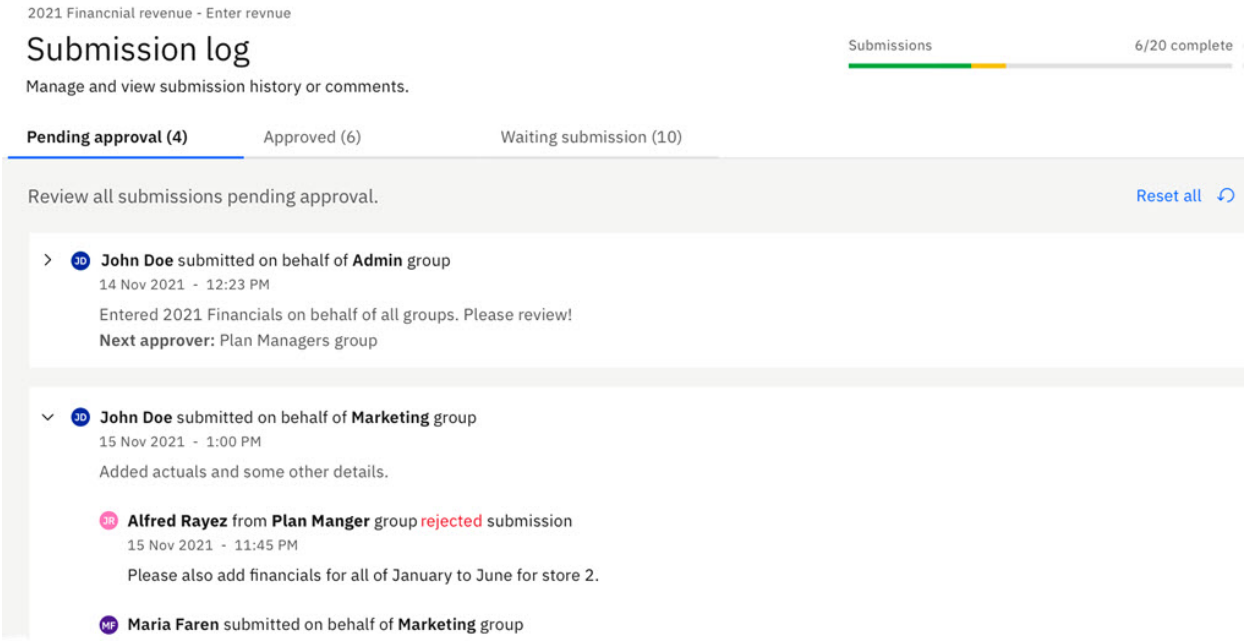



When no submission has been made for a step, the **Submissions** line does not include a status bar.

The yellow indicator in the status bar represents the percentage of submissions that are pending approval. When all submissions for a step are approved, the status bar appears dark green.

- Each step includes a link to the **Submissions** page. You can click the link to view the submission log for the step. The log captures the history and comments for all submissions. You can use the **Pending**

**approval**, **Completed**, and **Waiting submission** tabs to focus on submissions in any of those labeled states.




Plan owners can view all log entries while approvers can see only log entries for contributor groups they have been assigned to. Contributors can see only the log entries that apply to the group to which they belong.

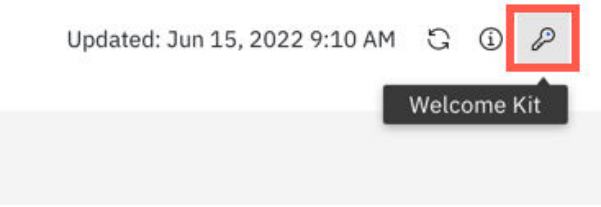
## Reset password for Remote Desktop user accounts (Cloud only)

Subscription administrators can now reset passwords for Remote Desktop user accounts from the new **Welcome Kit settings** page under **Administration**. Password reset can be done for a specific or all Remote Desktop users in an environment.

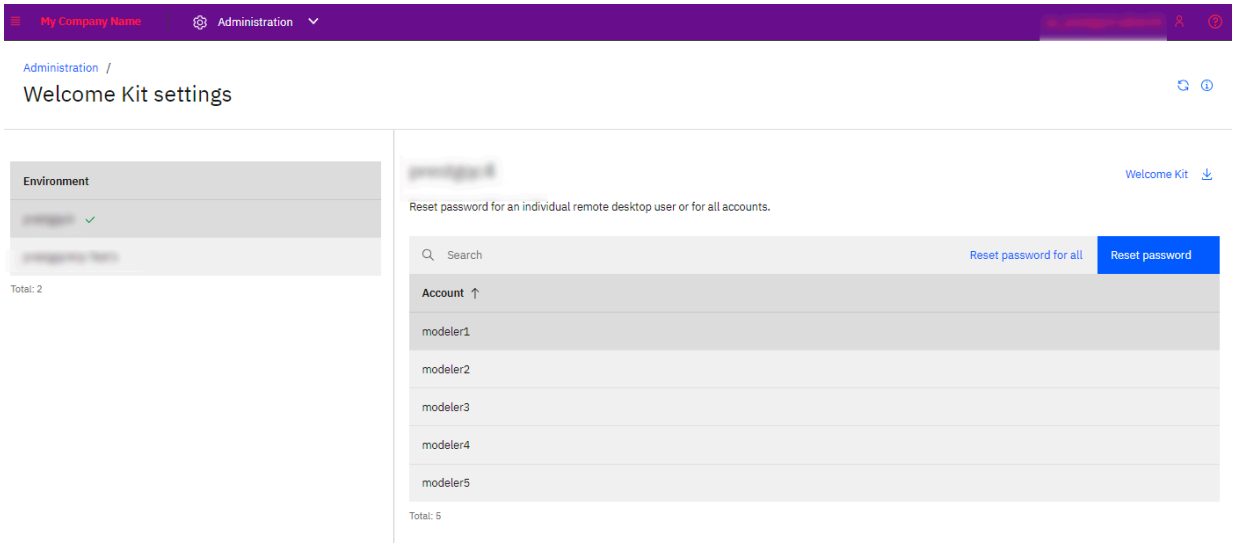
**Important:** This password reset feature applies only to Remote Desktop user accounts that use rich tier components to access TM1 and does not apply to Planning Analytics Workspace users.

To reset password for a Remote Desktop user:

1. On the **Home** page, click **Administration**.
2. Click the **Welcome Kit** icon in the toolbar.




3. On the **Welcome Kit settings** page, select an environment.
4. From the list of modelers, select the user for whom you want to reset the password and click **Reset password**.

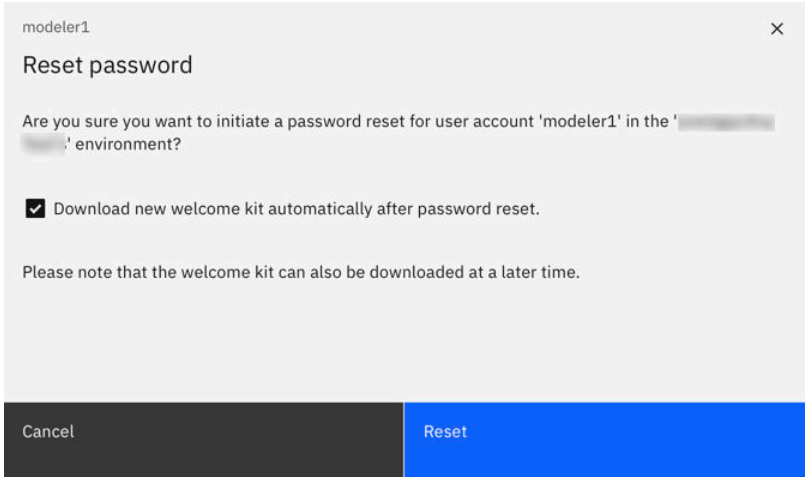



If you want to reset the password for all users in that environment, select **Reset password for all** instead.

5. A **Reset password** window opens.

   When you reset a password, a new welcome kit automatically downloads with the new password.

   If you are resetting passwords for more than one user (but not for all users) and do not want the welcome kit to download automatically upon each password reset, deselect the download option.




   You can download the welcome kit after the password is reset for all your selections.

6. Click **Reset** to reset the password.

**Note:** You cannot reset the password for a user for whom a password reset is in process. However, you can reset password for other users in the same or a different environment.

## Updated menus for rows, columns, and data cells in views

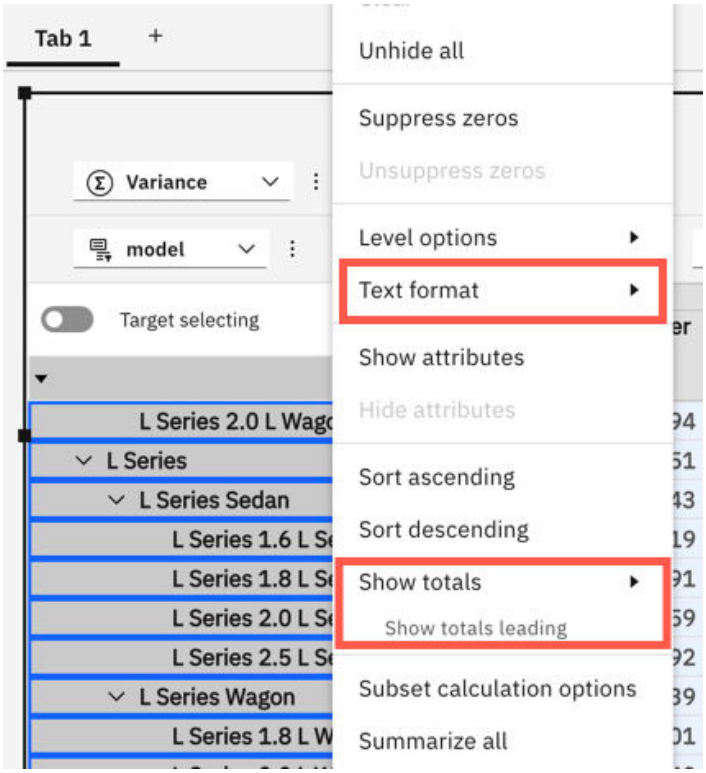
As part of the ongoing effort to standardize the Planning Analytics Workspace user experience, the menus for rows, columns, and data cells in the new experience view are now updated.

There is no change in how the menu options function but the new design includes usability improvements. A number of menu options were renamed:

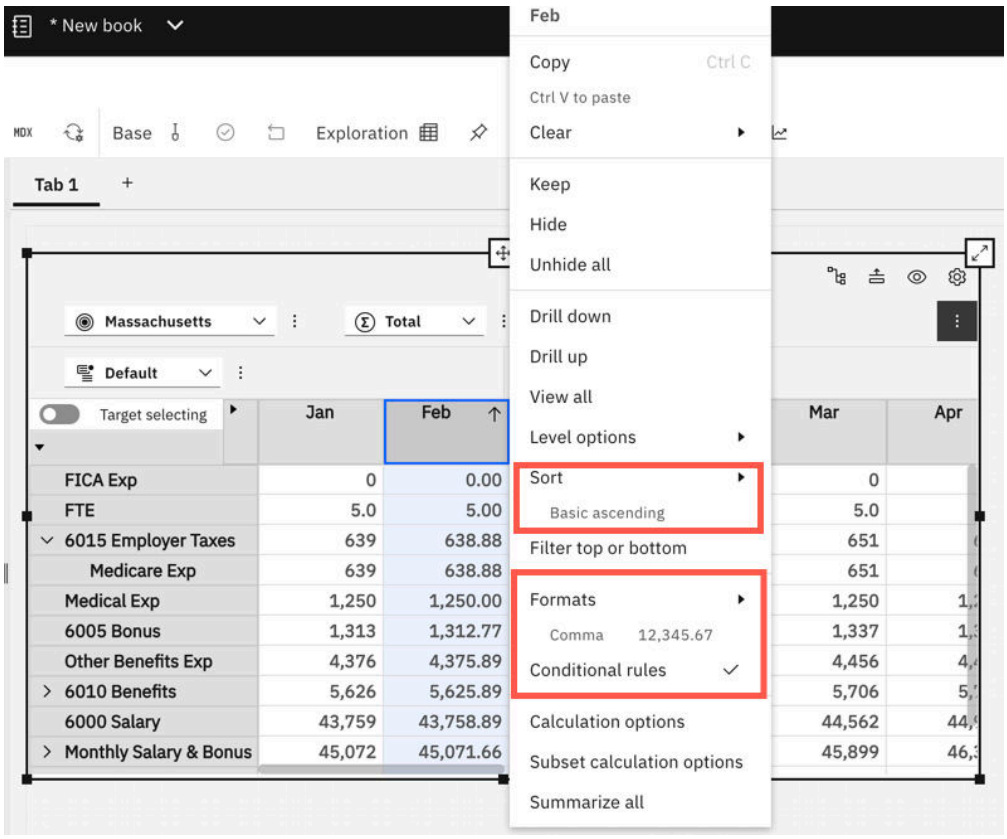
- **Create calculations** is now **Calculation options**
- **Format values** is now **Format**
- **Conditional format** is now **Conditional rules**

Several menu options are now grouped so you can easily see all related options together. For example:

- All sorting options are now under **Sort**
- Word wrapping options (for Headers only) are under **Text format**
- Show totals options are under **Show totals**



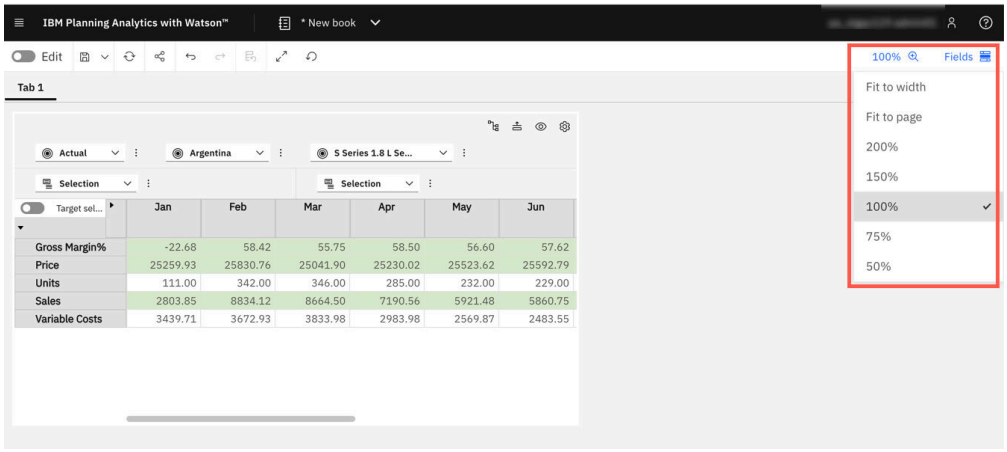

Some grouped options also include hints within the menu allowing you to quickly see the selections that apply to the row, column, or data cell.




## Adjust page size in books

You can now adjust page size in books with the new zoom feature. Select a preset size to see your content at that magnification level. You can also resize the page view to fit the page or fit to width of your window. In both cases, the page size is responsive and automatically adjusts to fit the page or window when you resize your window.

**Note:** The zoom feature works only in consumption mode.

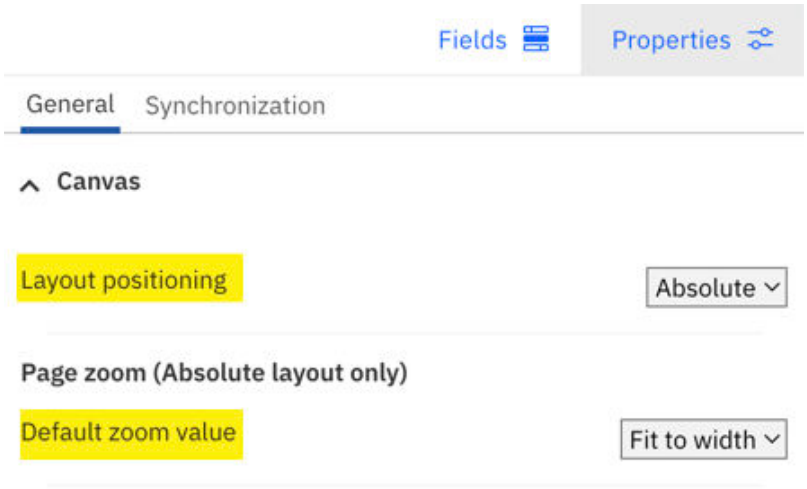



To use the zoom feature, make sure that the book is in **Absolute** layout. The zoom feature is not supported in books that use **Relative** layout position.

**Note:** When you upgrade from classic experience to new experience, depending on how you added tabs in books, the zoom feature might not work correctly. For more information, see known issue Zoom does not work correctly in upgraded books with templates.

You can change the layout position of a book in **Edit** mode on the **General** tab under **Properties**.

You can also set a default zoom value for your book under **Properties** so that it always opens in the selected size.

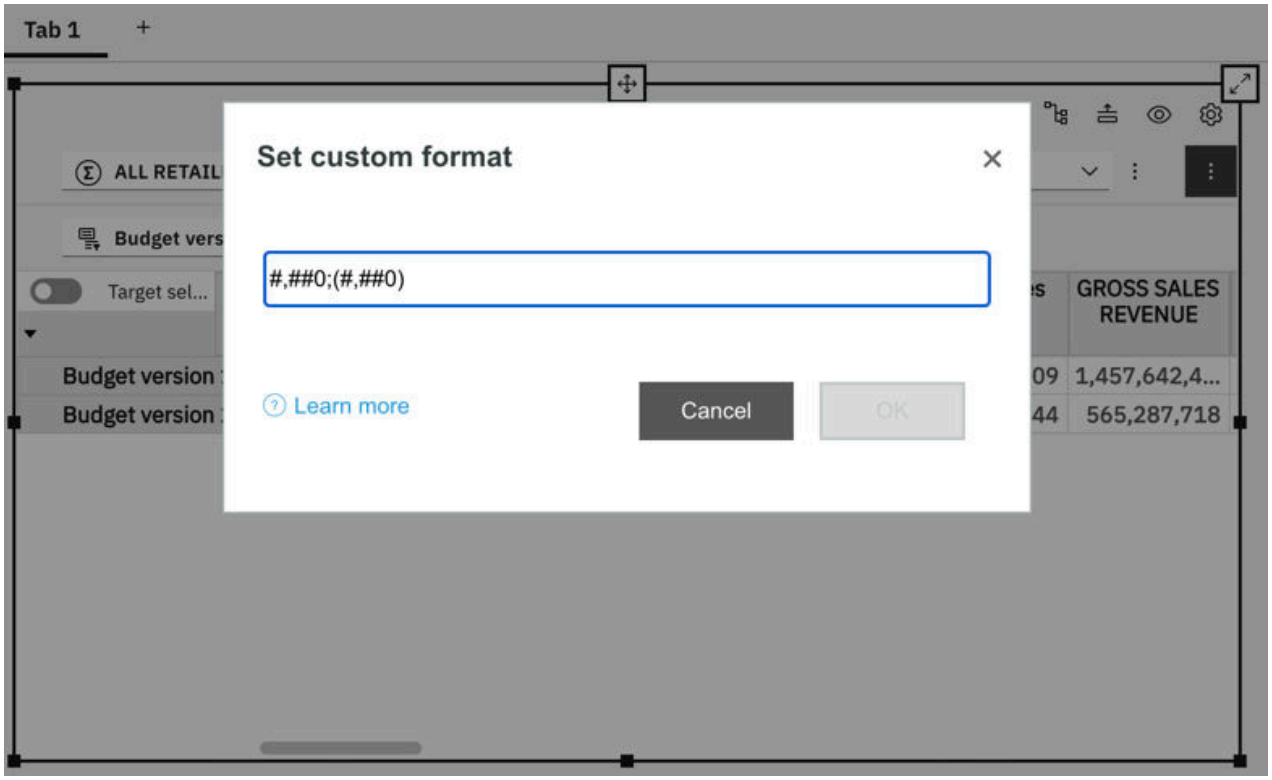



When you enter full screen mode, your zoom value is overridden and the page size changes to **Fit to page** to allow you to see the full contents of your page. However, you can still change your zoom value in full screen mode. When you exit full screen mode, you automatically go back to the previously selected zoom value.

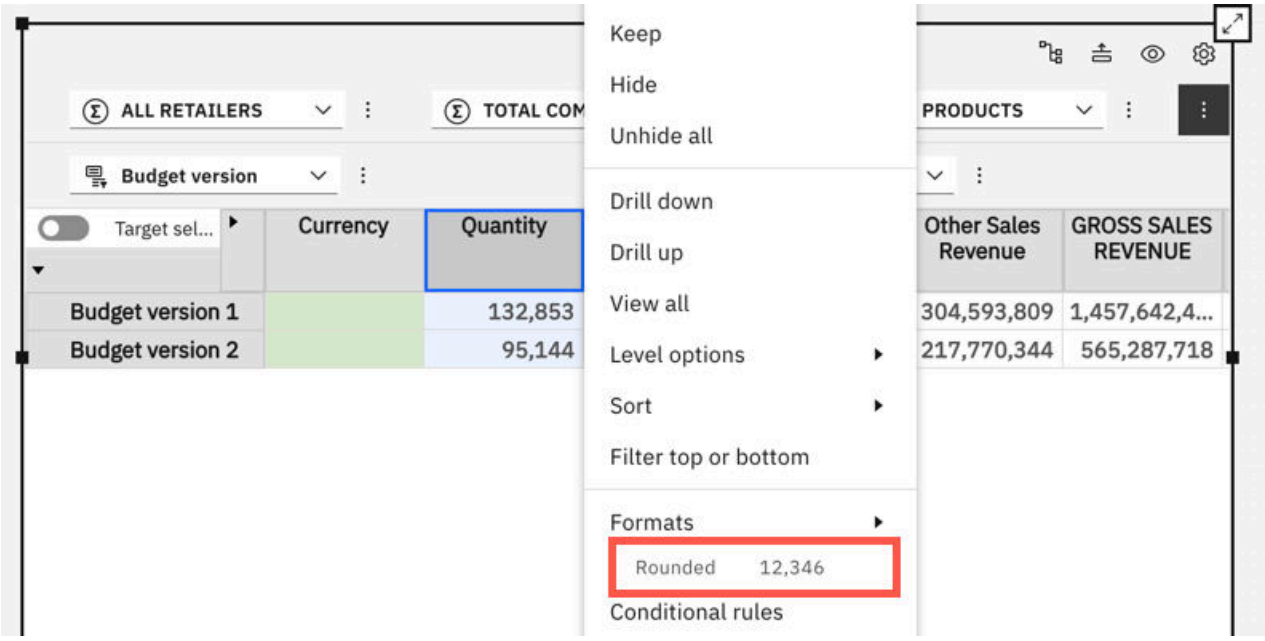
## Enhanced data format feature

An enhancement to the data format feature in views now lets you quickly see the data format that was applied on a row, column, or an entire view.

When you apply a built-in data format or custom format to a row, column, or the entire view, the pattern for that format is now retained in the **Set custom format** field under **Custom format**.




For built-in data formats, you can also see the applied format under **Formats** when you right-click the relevant row or column.

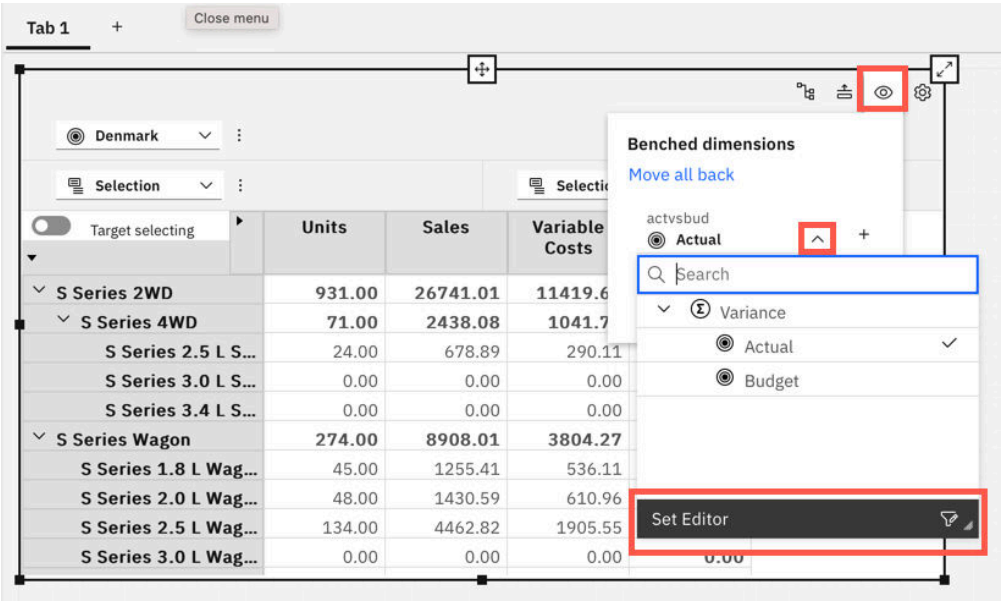



For more information, see Setting the data format in a view.

## Open set editor for a benched dimension

In new experience views (or explorations), you can now open the set editor for dimensions that are on the bench.

To open the set editor for a benched dimension, click the **Bench** icon in a view and double-click the dimension. You can also click the drop-down menu icon for the benched dimension and open the set editor from the menu.




## Provide a direct path to a Planning Analytics Workspace folder

You can now share a URL to your tabs that are labeled: **Shared**, **Personal**, **Favorites**, and **Recents**. You can also share a URL to provide a direct path to a folder within your tabs.

When you select tabs or select folders, the URL changes in your browser. The URL also changes in your browser when you select the breadcrumb links that are displayed under the tabs.

Copy and paste the updated URL to share a direct path to a tab, folder, or breadcrumb link.

## Update to Planning Analytics Workspace Local installation on Windows Server 2016

In Planning Analytics Workspace Local version 2.0.78 and later, the Docker service file is no longer bundled with the installation kit. In previous versions, Docker is a prerequisite for installing Planning Analytics Workspace Local on Windows Server 2016.

When you now install Planning Analytics Workspace Local on Windows Server 2016, the installation script looks for Docker and if an existing version is not found, displays an error. To resolve the error and continue the installation process, you need to download Mirantis Container Runtime.

Before Planning Analytics Workspace Local 2.0.78, the installation script used the Docker file that was bundled with Planning Analytics Workspace to continue installation if Docker was not found on your system.

For more information, see Install Planning Analytics Workspace Local on Windows Server 2016.

## Enable email notifications in Planning Analytics Workspace Local

Planning Analytics Workspace Local now supports both SMTP and SendGrid email services.

To enable the email service, specify the following parameters in the Planning Analytics Workspace Local configuration file. The configuration file is `paw.ps1` for Windows and `paw.env` for Linux. For more information, see Configure parameters.

**ENABLE_EMAIL**
Set this property to true to enable the email service in Planning Analytics Workspace Local. Either **EMAIL_SMTP_URL** or **SENDGRID_API_KEY** parameter must also be specified. If both parameters are specified, **EMAIL_SMTP_URL** is used.

Default: false

**EMAIL_SMTP_URL**
Use this parameter to enable the SMTP email service. The parameter specifies the SMTP server URL. The following are examples of such URLs:

```
smtp://username:password@smtp.example.com:587
```

```
smtp://username:password@smtp.example.com:587?secure=false
```

```
smtps://username:password@smtp.example.com:465
```

If you specify this parameter, the **SENDGRID_API_KEY** is not used, even if it's specified.

Default: empty (no URL is specified)

**SENDGRID_API_KEY**
Use this parameter to enable the SendGrid email service. The parameter specifies the API key for the SendGrid email account.

**Note:** You might need to contact the email administrator for your organization to obtain the API key that is specific to your email service.

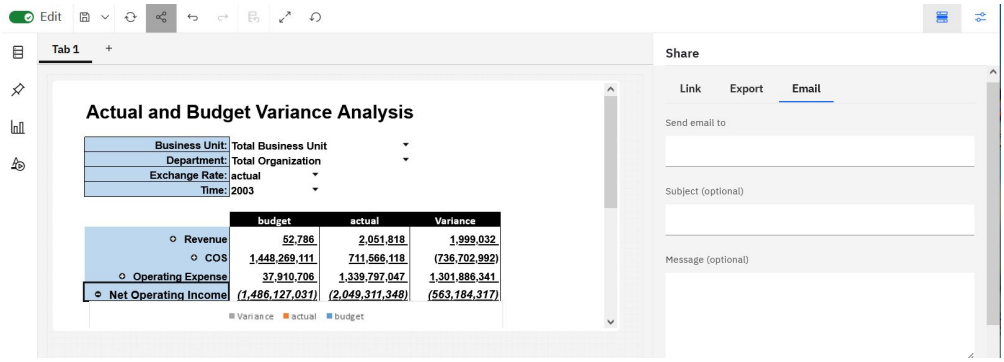
If you specify this parameter, specify also **EMAIL_FROM**, but don't specify **EMAIL_SMTP_URL**.

Default: empty (no API key is specified)

**EMAIL_FROM**
Specifies the sender address if it is required by the email server. This parameter is mandatory when **SENDGRID_API_KEY** is specified.

**Note:** When you change a paw configuration file, you must run `./scripts/paw` for your operating system or click **Start** in the Planning Analytics Workspace administration tool for your changes to take effect. Only services that are affected by the configuration change will restart.

The **Email** tab is now available in the user interface when you share a book or view.

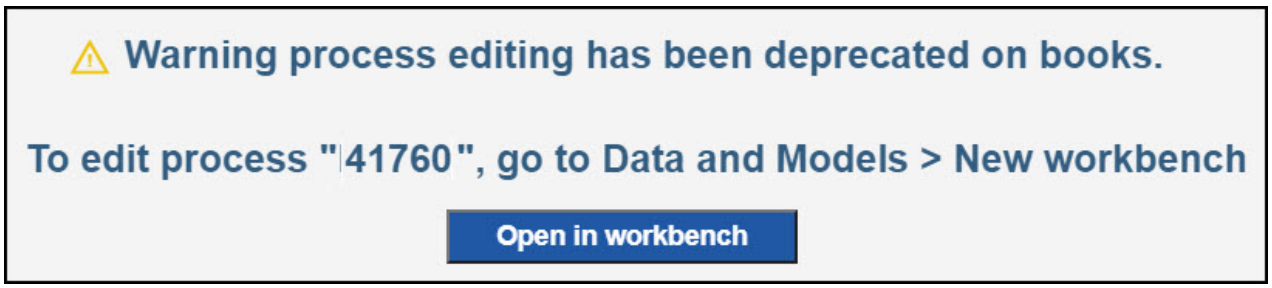



For more information, see Enable email notifications.

## Deprecation notice - process editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.79

Starting with Planning Analytics Workspace 2.0.79, processes must be created and edited on a modeling workbench.

If you are an administrator or modeler and you attempt to open a process in a book in 2.0.79, you'll receive a notification that process editing must be performed on a workbench. You can click **Open in workbench** to immediately open the process on a new workbench.




## Deprecation notice for IBM Planning Analytics Workspace Classic experience

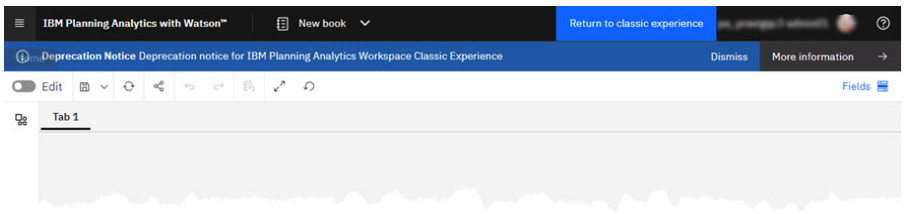
The IBM Planning Analytics team intends to remove the capability to remain on Planning Analytics Workspace Classic and preview the Planning Analytics Workspace new experience as part of the Planning Analytics Workspace 2.0.79 update in August 2022.

The 'new experience' is a major update to the Planning Analytics Workspace user experience released as part of the 2.0.56 update in October 2020. The Planning Analytics Workspace new experience provides an updated look and feel, improved navigation and content management. It also introduces Applications and Plans, a dedicated modeling environment, and Predictive Forecasting capabilities.

The user experience in 2.0.55 and earlier releases is referred to as Planning Analytics Workspace Classic. Cloud environments provisioned before the 2.0.56 update could remain on Planning Analytics Workspace Classic, preview the new experience, or permanently switch to the new experience.

Planning Analytics Workspace environments that use the Classic experience will automatically be updated to the new experience as part of the 2.0.79 update in August 2022. Planning Analytics cloud customers that use the Classic experience are encouraged to preview and switch to the new experience before August 2022.

If you are still using Planning Analytics Workspace Classic prior to the August 2022 update, administrators will see a deprecation notice banner on the home page when using Planning Analytics Workspace Classic or previewing the new experience. An administrator can click the **More information** link to view further details on the deprecation of Planning Analytics Workspace Classic.




The deprecation notice is presented to each administrator upon login. One administrator dismissing the banner dismisses the banner only for that administrator; other administrators can still see the banner.

The banner will reappear with each new Planning Analytics Workspace release until an administrator completes the switch to the new experience.

**Note:** This announcement does not impact Planning Analytics on-premise deployments.

## Known issues

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

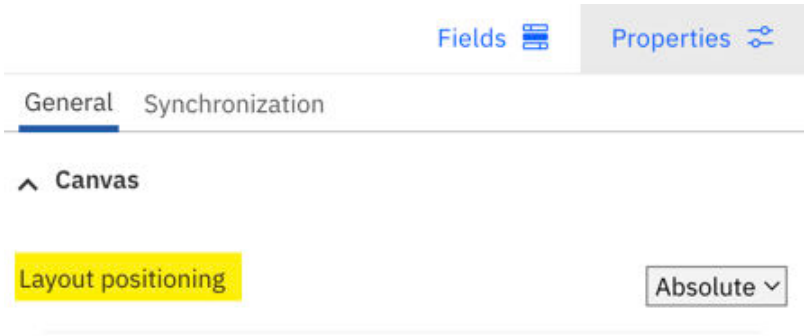
### Zoom does not work correctly in upgraded books with templates

When you upgrade from the classic experience to new experience, depending on how you added the tabs in books with templates, the zoom feature might not work correctly.

In classic experience, different tabs in a book can have different layouts (relative or absolute). However, in new experience books, you can have only one layout.

When you upgrade from classic to new experience, the layout of each tab is retained. Even though the book's layout might display as **Absolute**, tabs that are created with templates are relative layout. When zoom feature's **Fit to page** is applied to the relative tabs, a white margin appears around the book. If you select a zoom value that is greater than 100%, a scroll bar appears on the relative tabs.

To fix this issue, change the layout position of the book from **Absolute** to **Relative** under **Properties** and save the book. Then, change the layout back to **Absolute**. The zoom feature will now work correctly.




# 2.0.77 - What's new, June 2, 2022

Learn about new features and known issues in version 2.0.77 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

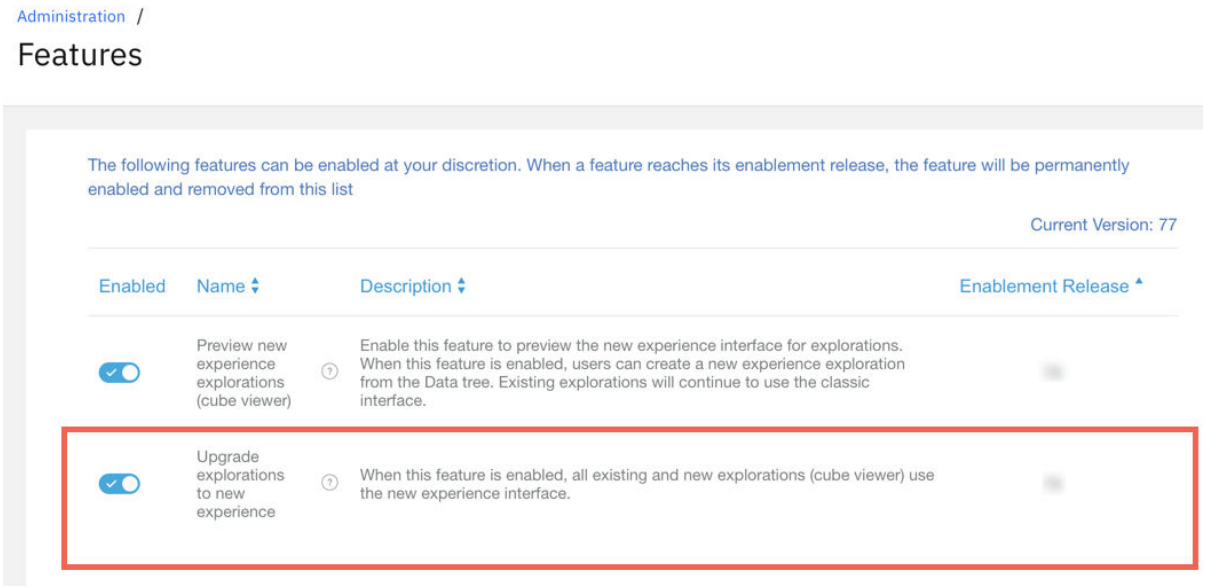
### Timeline of previous Planning Analytics releases

For a comprehensive listing of Planning Analytics component releases by month and year, see Timeline for IBM Planning Analytics Releases. You can view versions and release months for the following components:

- Planning Analytics Local TM1 Server
- Planning Analytics Workspace
- Planning Analytics for Microsoft Excel
- Planning Analytics Spreadsheet Services/TM1 Web

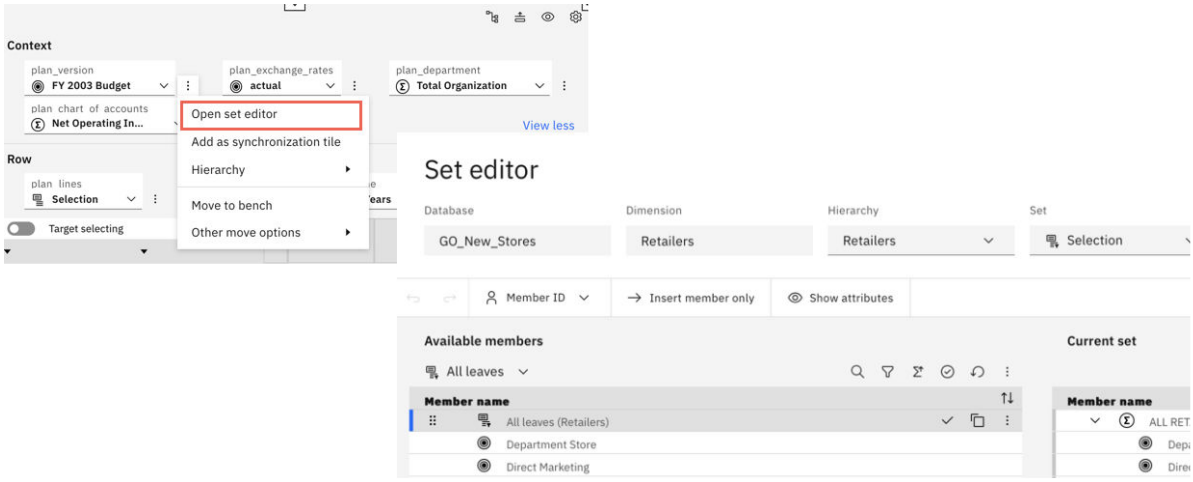
## Upgrade explorations to the new experience

An option to upgrade explorations (cube viewer) to the new experience is now available on the **Features** administration tab. The new experience interface offers numerous enhancements and improvements.

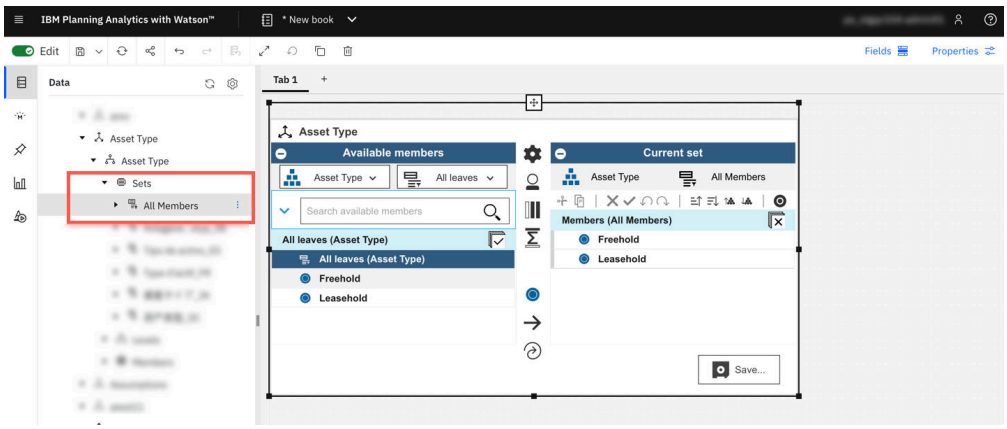



When you enable this feature, all existing and new explorations in books use the new experience interface. Explorations that previously used the classic interface are converted to new experience but retain the look (colors and font) of the classic interface.

Upgrading explorations to the new interface also upgrades set editors but only those that are opened from the new experience exploration.




If you add a set editor as a widget to a book from the data tree, the set editor continues to use the classic interface.




**Note:** Enabling the upgrade feature, upgrades only the explorations in Planning Analytics Workspace books. Explorations and set editors in the modeling workbench and IBM Planning Analytics for Microsoft Excel are not changed.

A preview of the new experience exploration and set editor was released in Planning Analytics Workspace 2.0.71 and is also available on the **Features** administration tab. You can enable preview to explore the new experience interface and compare the new explorations with the classic ones before you upgrade.

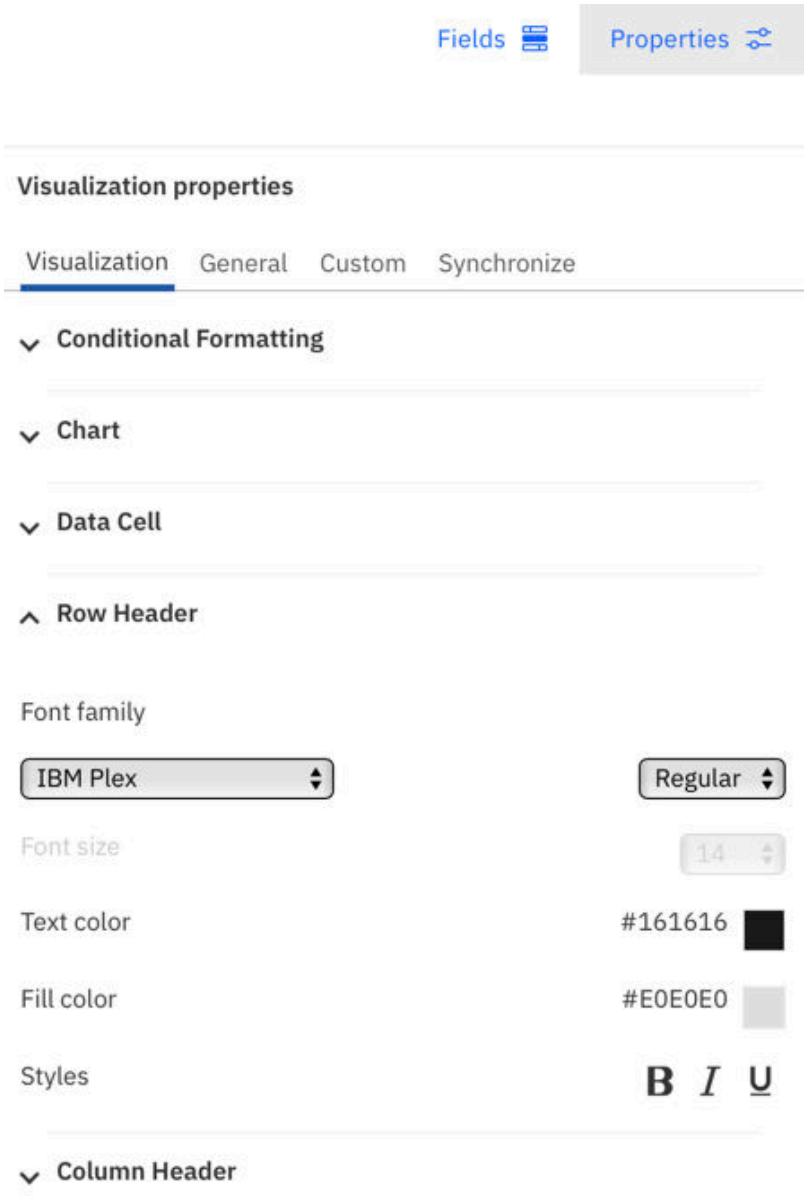
For more information, see New experience view and set editor.

## Changes to text properties for explorations

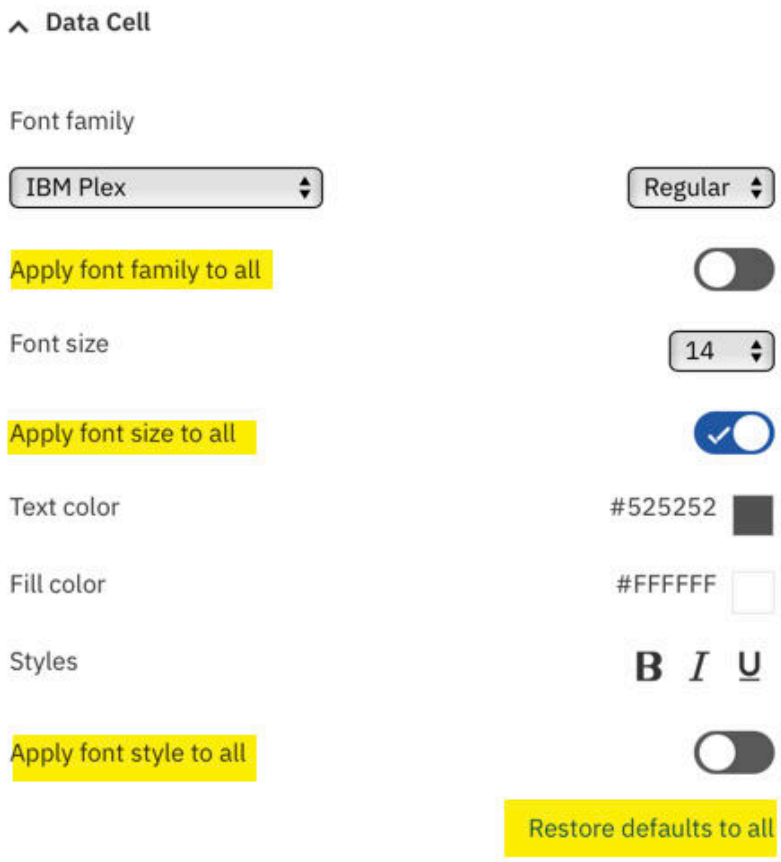
The text properties for explorations are now under the **Visualization** tab instead of the **Custom** tab in **Properties**.

**Note:** Text properties for single cell widgets continue to be under **Custom** tab.

The **Data Cell**, **Row Header**, and **Column Header** sections under the **Visualization** tab each have their own text properties. The following example shows the text properties for the **Row Header** in an exploration.



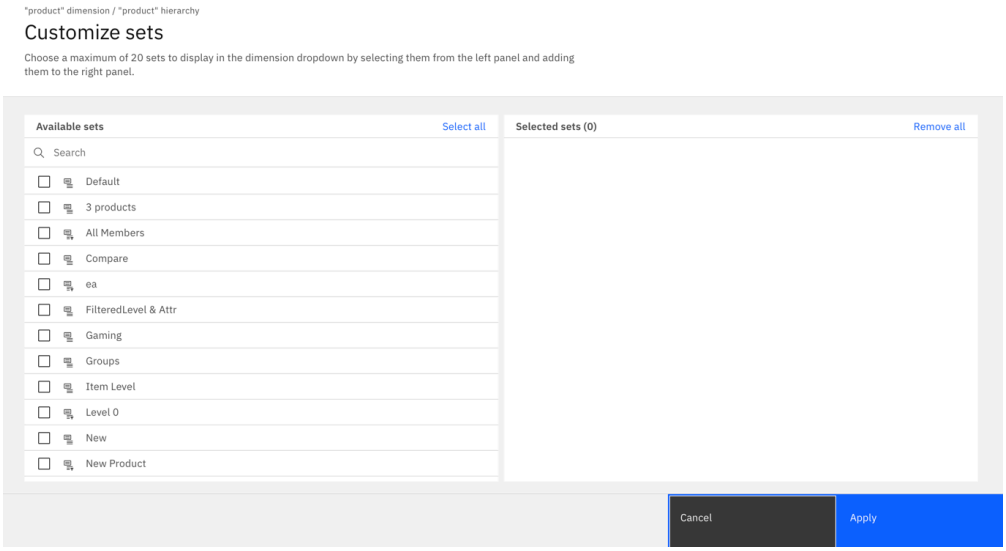

You can quickly apply the font family, font size, and font style to all sections or restore defaults for all sections under **Data Cell**.



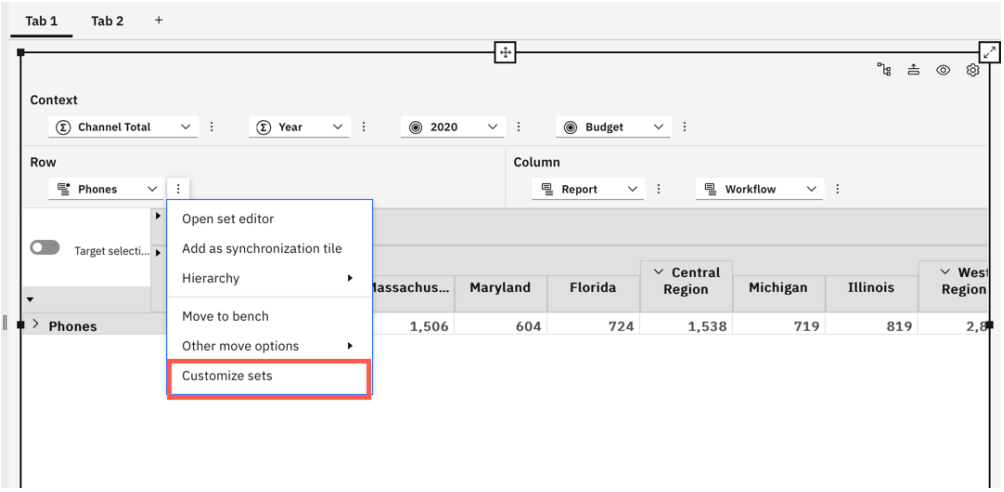

## Select the sets that display in a dimension drop-down

Use the new **Customize sets** option in the new experience exploration (cube view) and visualizations to limit the number of sets that appear in a dimension drop-down. You can select up to 20 sets for a dimension.

**Note:** The **Customize sets** option does not limit the sets that you see in the set editor. You can still see the full list of sets in the set editor.



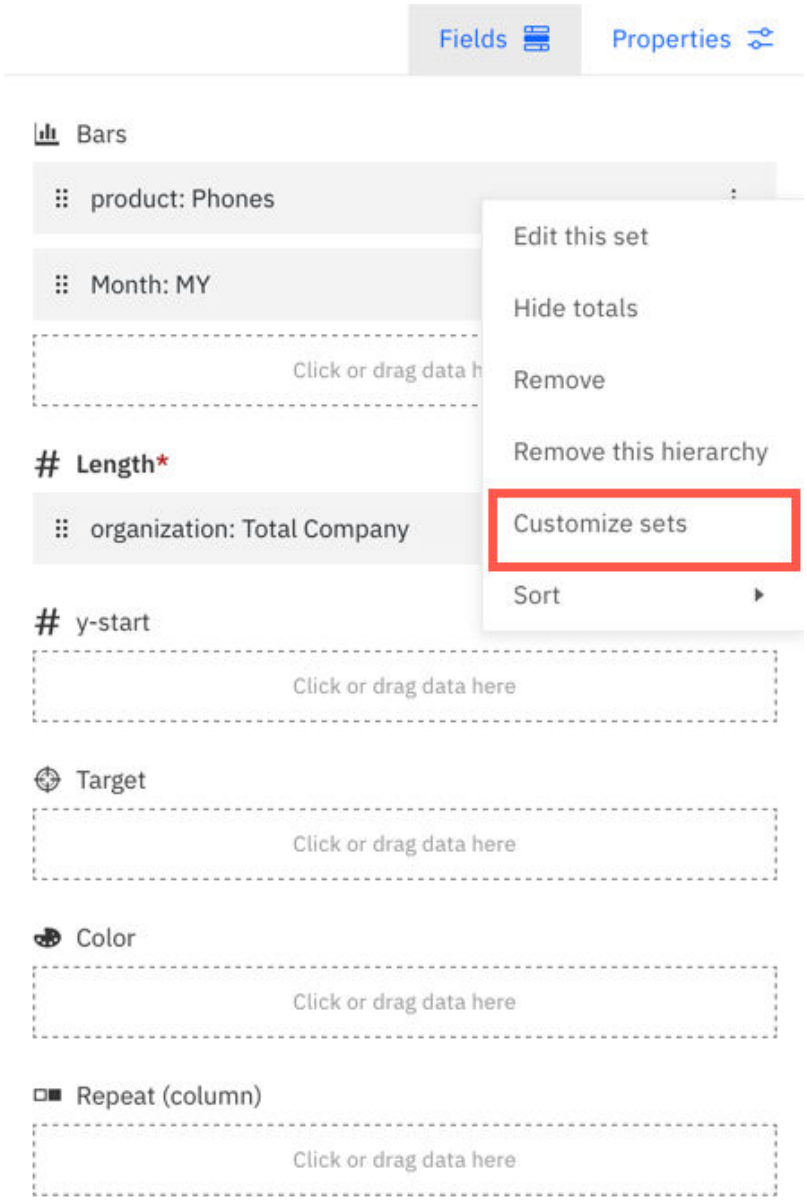

To **Customize sets** in an exploration, click the context menu of a dimension in the **Row** or **Column** area. You cannot customize sets for dimensions in the **Context** area.




You can move dimensions with customized sets between the **Row**, **Column**, and **Context** areas without losing the limited sets.

However, if you move a dimension with customized sets off the exploration and add it to the book as a widget, the limited list of sets is lost. You will see all the sets for that dimension in its drop-down.

You can also customize sets for dimensions in visualizations. Select the visualization in **Edit** mode and click **Fields**. Click the menu for the dimension you want to customize, and select **Customize sets**.

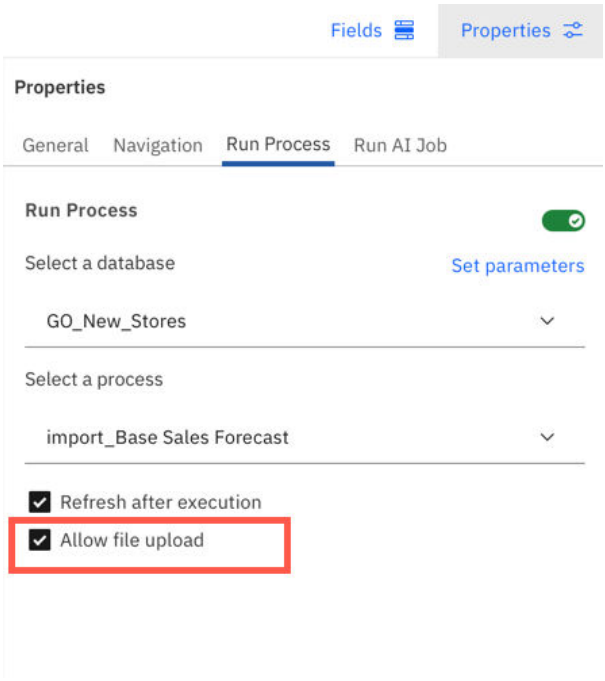



You can switch between a visualization and an exploration (or vice versa) without losing any customized sets that you applied.

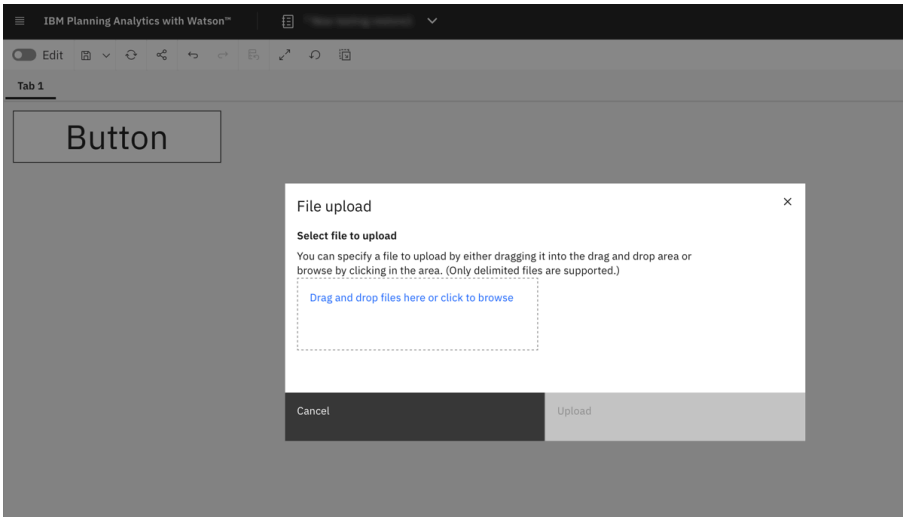
For more information, see Customize sets in dimension lists.

# Upload files to a Planning Analytics database with the action button

In Planning Analytics Workspace books, the Action Button has a new property, **Allow file upload**, that lets you upload a file to a Planning Analytics database and run a process to use that file. This can be helpful if you want to, for example, import data from a specific file into a view you are using.

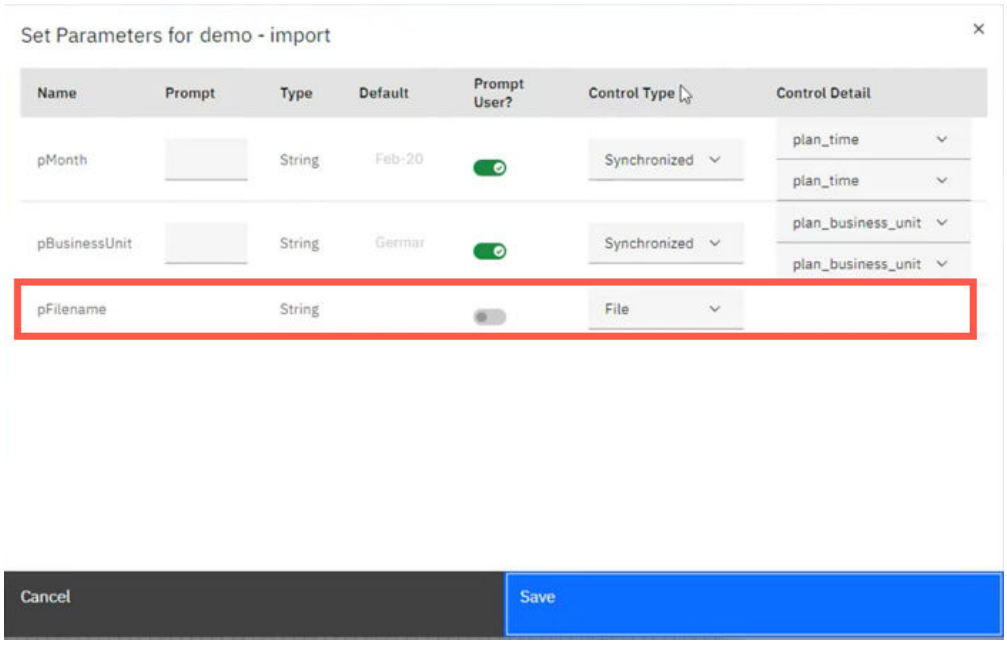



When you configure an action button to run a process (under **Properties > Run Process**), selecting **Allow file upload** prompts you to upload a file when you click the Action Button.




**Note:** The process runs only if the file is uploaded successfully.

A new **Control Type** parameter, **File**, is now also available under Set parameters. This new parameter displays only when the **Allow file upload** property is selected.




The **File** control type passes the file name of the uploaded file to the process parameter.

For more information, see Upload files to a database with the action button.

## Manage files with File manager

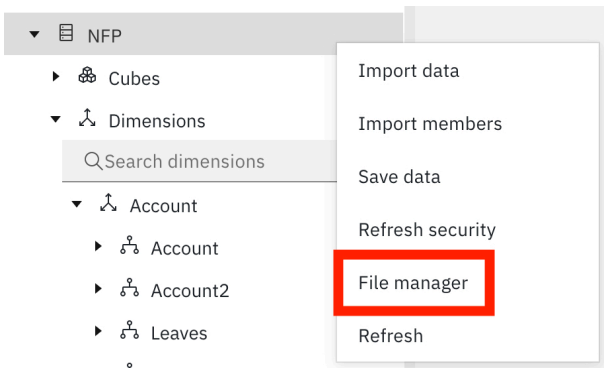
A new tool, **File manager**, shows a list of all the files and folders that have been uploaded to the Planning Analytics database and allows you to manage those items.

**File manager** is available only on the modeling workbench; you cannot open **File manager** from a book. You must be an Administrator or Modeler to work with **File manager**.

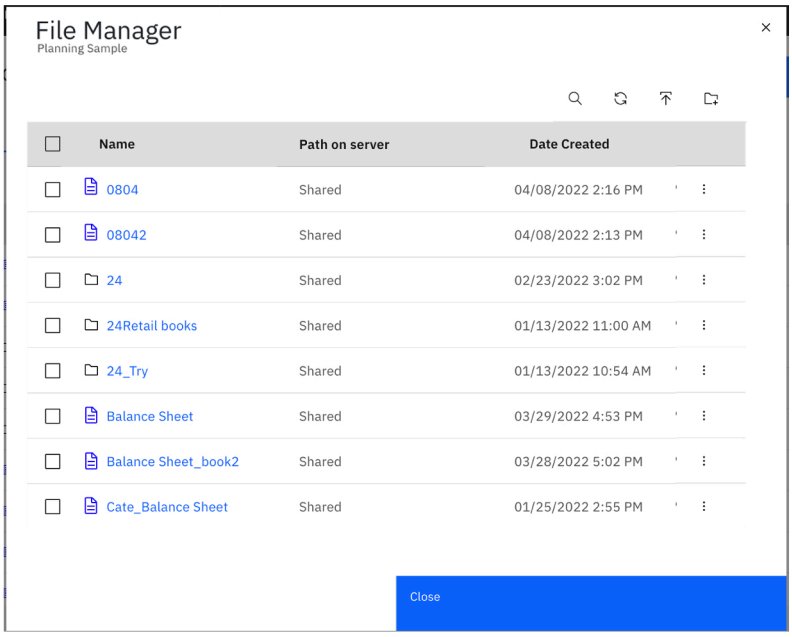
With **File manager**, you can complete the following tasks:

- Upload a file, or multiple files at once
- Delete a file, or multiple files at once
- Create folders
- Organize uploaded files into folders
- Delete folders
- Download files
- Refresh files

To launch **File manager** for a database, right-click the database in the Databases tree on a modeling workbench, then click **File manager**.




**File manager** displays a list of all the files and folders that have been uploaded to the server.

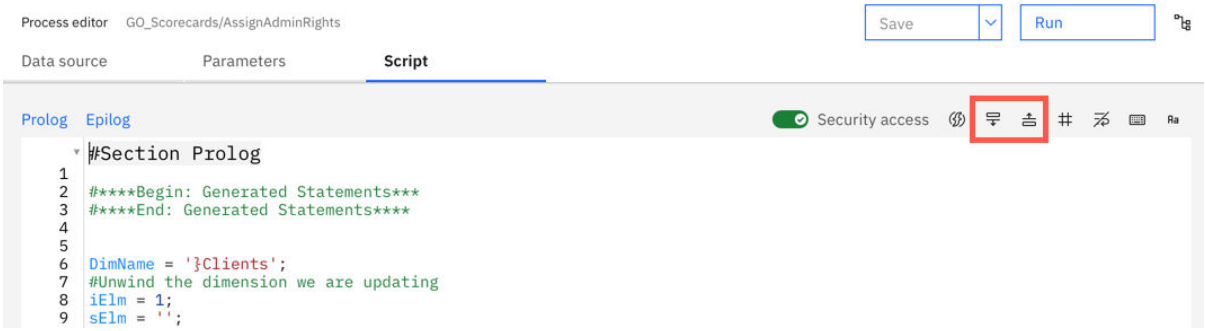



You can use the toolbar and the **Actions** button ⋮ to work with files and folders.

## Expand and collapse regions in the process and rules editor

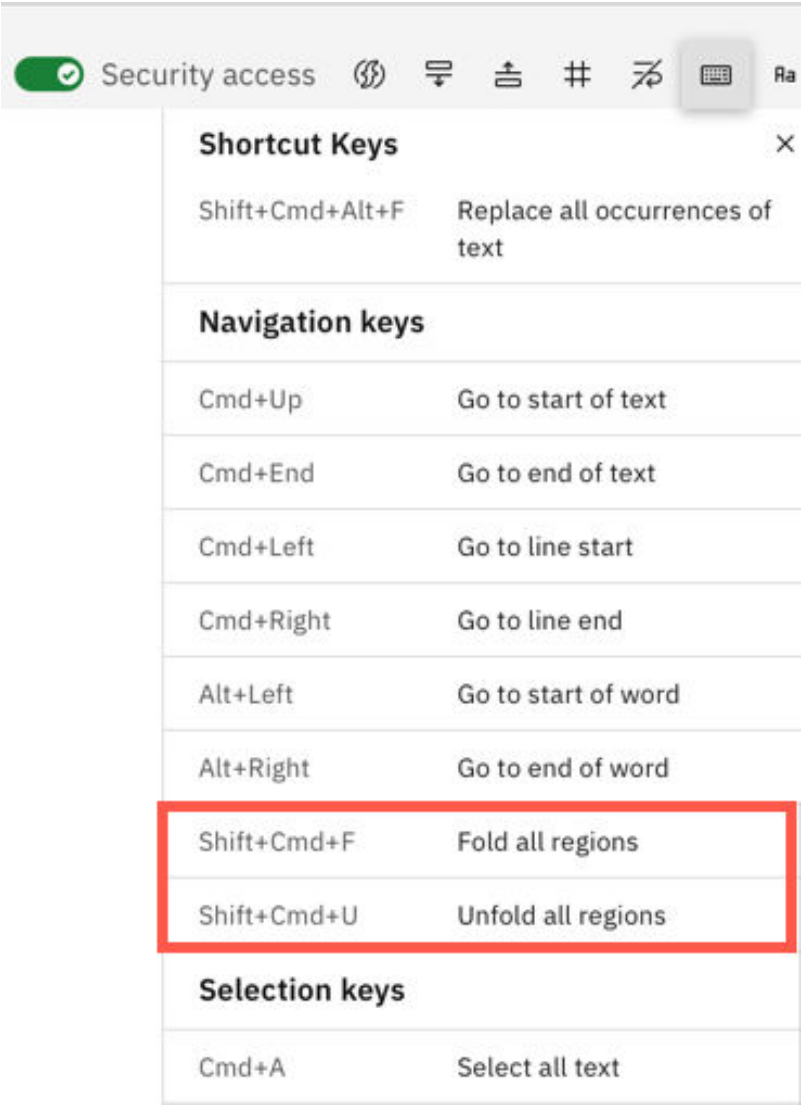
You can now expand and collapse regions in the process and rules editor on the modeling workbench. Before Planning Analytics Workspace 2.0.77, you could only expand and collapse regions one at a time.

Click **Unfold all regions** to expand all regions and **Fold all regions** to collapse regions in both the process and rules editor.




In the process editor, folding and unfolding regions starts at the **Section** level.

Two new shortcut keys for the expand and collapse features are now also available in the process and rules editor.

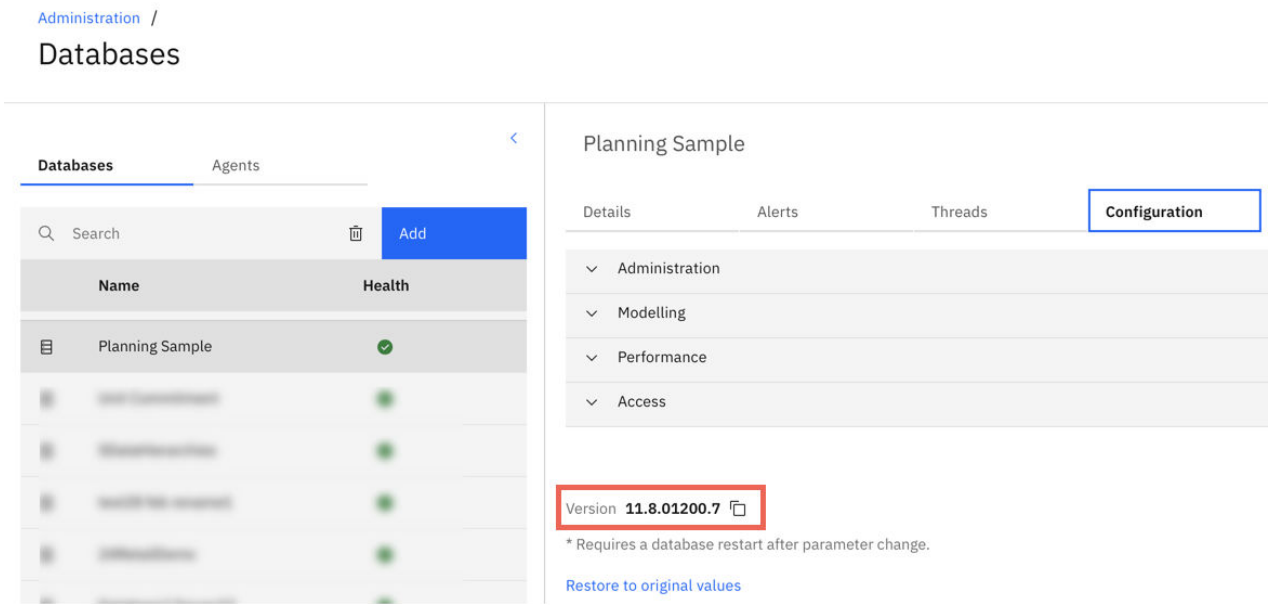



## Viewing the TM1 database version (admins only)

You can now see the version of a TM1 database in **Administration** under the **Home** page in Planning Analytics Workspace.

To see the TM1 database version:

1. On the **Administration** page, click the **Databases** tile.
2. On the **Databases** page, select a database and click **Configuration**.

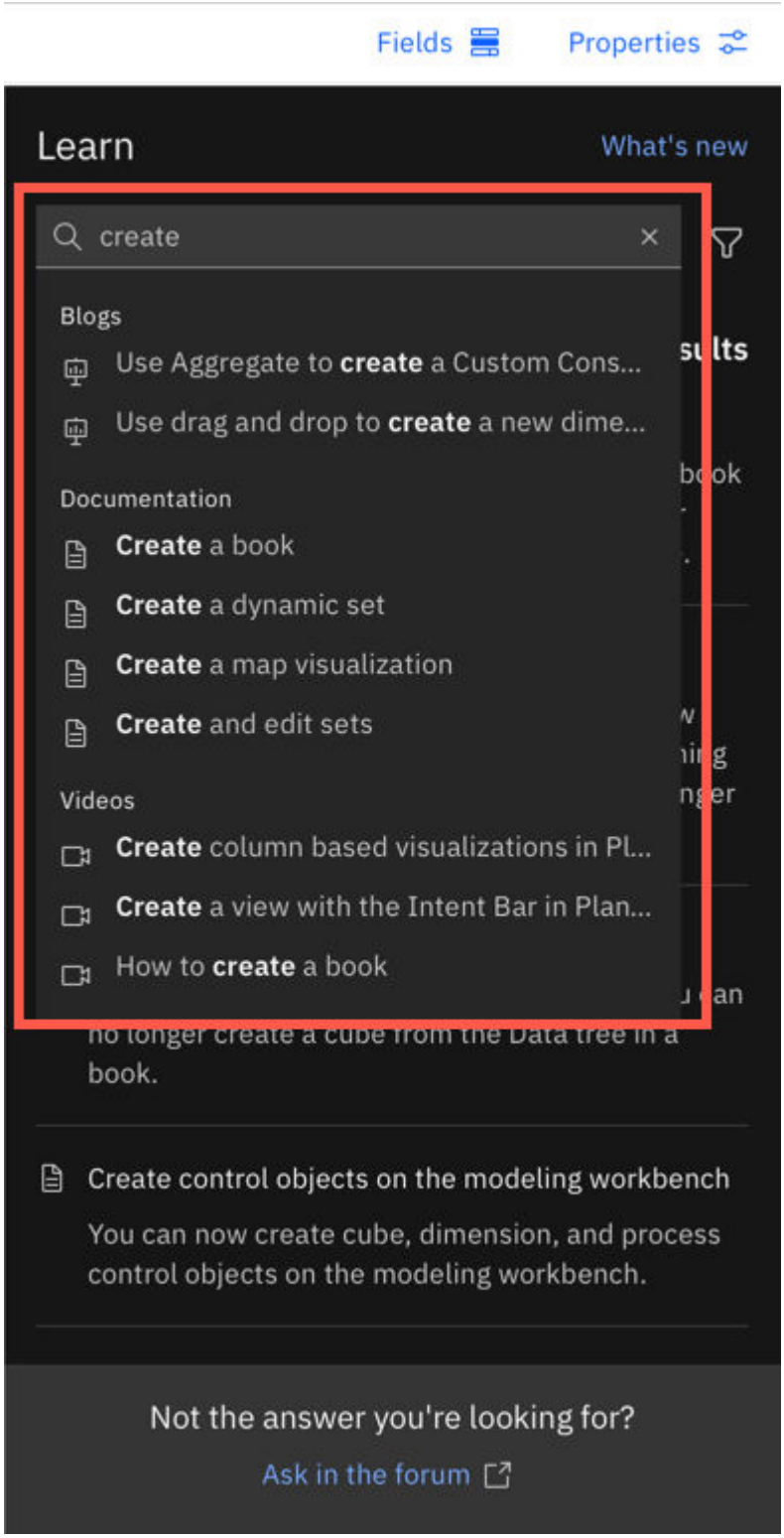



If you need to copy the version number, click the **Copy** icon next to the version number.

## Auto-complete and popular search suggestions in the cognitive Learn pane

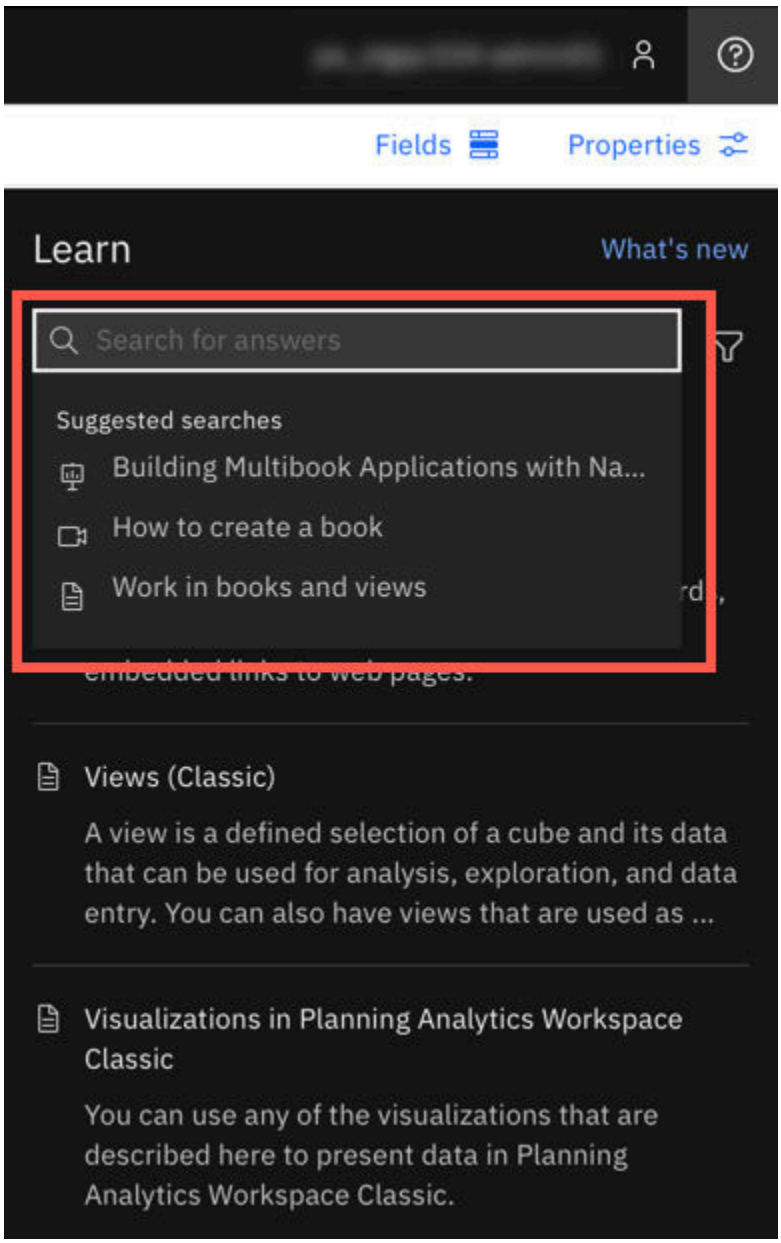
Find answers to your Planning Analytics Workspace questions faster! The Learn pane now predicts your search query as you type keywords in the **Search** field and displays a list of search suggestions from all content types.

This auto-complete feature makes it easy to complete your search quickly. Simply select a search suggestion to see related help content.




When you search in any supported language other than English, the auto-complete feature returns translated documentation in your search suggestions. You also see blogs and videos that match your search query, however, blogs and videos aren't translated.

Not sure what to search for in the Learn pane? Click the **Search** field to see a list of popular search suggestions.




For more information, see Find information.

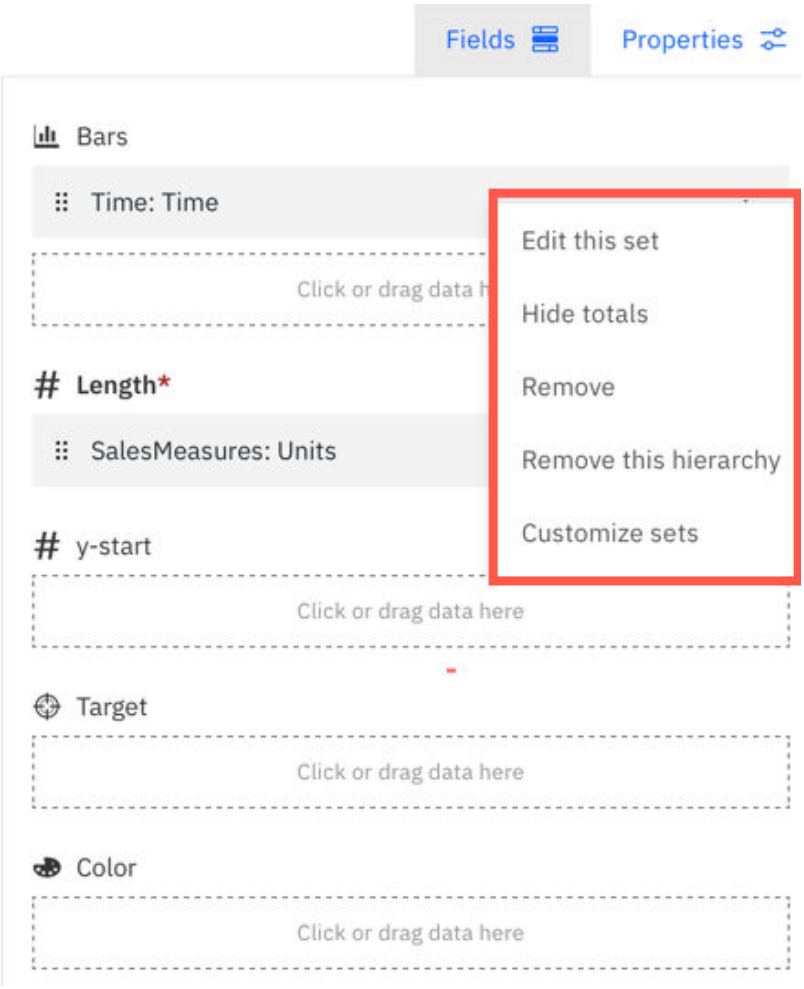
## Updated menu options for visualization field slots

The options in the menu for field slots under the **Fields** tab have been updated in Planning Analytics Workspace 2.0.77.

The following changes were made to the menu options:

- The **Hide from view** option was replaced with **Remove**.

  The **Remove** option lets you remove a dimension from the field it was assigned to and move it to the **Filters** section.
- **Remove this hierarchy** now moves the dimension from the field slot to the **Filters** section and reverts the set to the dimension's default set.
- The **Remove field** option is no longer available.




To see the menu for a field slot:

1. While in **Edit** mode, select the visualization in your book.
2. Click **Fields**.
3. Click the menu icon in a field slot to see the options for that field.

For more information, see Create a visualization directly in a book.

## Hide hierarchy name to shorten visualization titles

You can now hide the hierarchy name in an axis or legend title to shorten the labels in a visualization. The axis and legend title labels show both the hierarchy name and the name of the set or member that was selected. When the hierarchy name is hidden, you see the name of only the set or the selected member.

To hide the hierarchy name in a title:

1. Select a visualization in **Edit** mode.
2. Click **Properties**.
3. To hide the hierarchy name in an axis title, expand the **Axis** section under **Visualization**.

   To hide the hierarchy name in a legend title, expand the **Legend** section.
4. Enable **Hide hierarchy name**.

   For visualizations with axes, you can hide the hierarchy name in the **Item axis title**, **Value axis title**, or both.

**Note:** The **Hide hierarchy name** toggle works only on automatically generated titles and does not work on custom titles.

You can still see the full labels for axes or legend titles in the respective field slots on the **Fields** tab.

If multiple dimensions are assigned to a field in a visualization with axes, hiding the hierarchy name shortens the title for each dimension.

Similarly, for visualizations that have multiple legends, enabling **Hide hierarchy name** shortens the title for all legends in the visualization.

## Restore a book to the currently open tab

When you reset a book from a saved tab to reverse changes you made, the book now restores to that currently open tab. Before Planning Analytics Workspace 2.0.77, **Reset book** restored the book to the first tab.

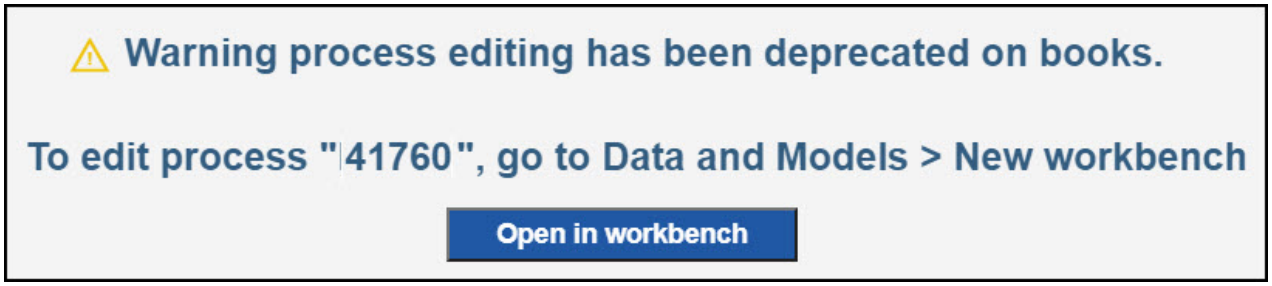
If you reset a book from a tab that has not been saved, the book returns to the first tab.

For more information, see Resetting a book.

## Deprecation notice - process editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.79

Starting with Planning Analytics Workspace 2.0.79, processes must be created and edited on a modeling workbench.

If you are an administrator or modeler and you attempt to open a process in a book in 2.0.79, you'll receive a notification that process editing must be performed on a workbench. You can click **Open in workbench** to immediately open the process on a new workbench.




## Deprecation notice for IBM Planning Analytics Workspace Classic experience

The IBM Planning Analytics team intends to remove the capability to remain on Planning Analytics Workspace Classic and preview the Planning Analytics Workspace new experience as part of the Planning Analytics Workspace 2.0.79 update in August 2022.

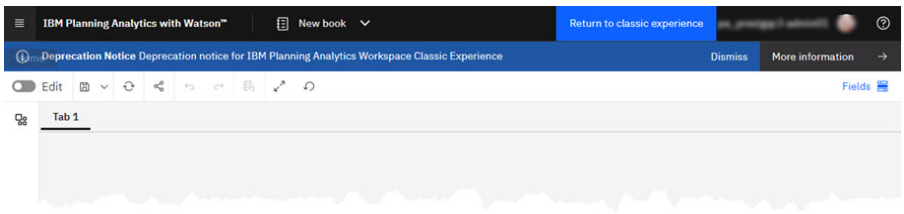
The 'new experience' is a major update to the Planning Analytics Workspace user experience released as part of the 2.0.56 update in October 2020. The Planning Analytics Workspace new experience provides an updated look and feel, improved navigation and content management. It also introduces Applications and Plans, a dedicated modeling environment, and Predictive Forecasting capabilities.

The user experience in 2.0.55 and earlier releases is referred to as Planning Analytics Workspace Classic. Cloud environments provisioned before the 2.0.56 update could remain on Planning Analytics Workspace Classic, preview the new experience, or permanently switch to the new experience.

Planning Analytics Workspace environments that use the Classic experience will automatically be updated to the new experience as part of the 2.0.79 update in August 2022. Planning Analytics cloud customers that use the Classic experience are encouraged to preview and switch to the new experience before August 2022.

If you are still using Planning Analytics Workspace Classic prior to the August 2022 update, administrators will see a deprecation notice banner on the home page when using Planning Analytics

Workspace Classic or previewing the new experience. An administrator can click the **More information** link to view further details on the deprecation of Planning Analytics Workspace Classic.




The deprecation notice is presented to each administrator upon login. One administrator dismissing the banner dismisses the banner only for that administrator; other administrators can still see the banner.

The banner will reappear with each new Planning Analytics Workspace release until an administrator completes the switch to the new experience.

**Note:** This announcement does not impact Planning Analytics on-premise deployments.

# 2.0.76 - What's new, May 5, 2022

Learn about new features and known issues in version 2.0.76 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

You can also review all new features introduced in previous versions in the IBM Planning Analytics Workspace new feature summary table. You can use the table to search for new features across all releases by description. You can also sort new features by release, date, functional area, and description.

### Timeline of previous Planning Analytics releases

For a comprehensive listing of Planning Analytics component releases by month and year, see Timeline for IBM Planning Analytics Releases. You can view versions and release months for the following components:

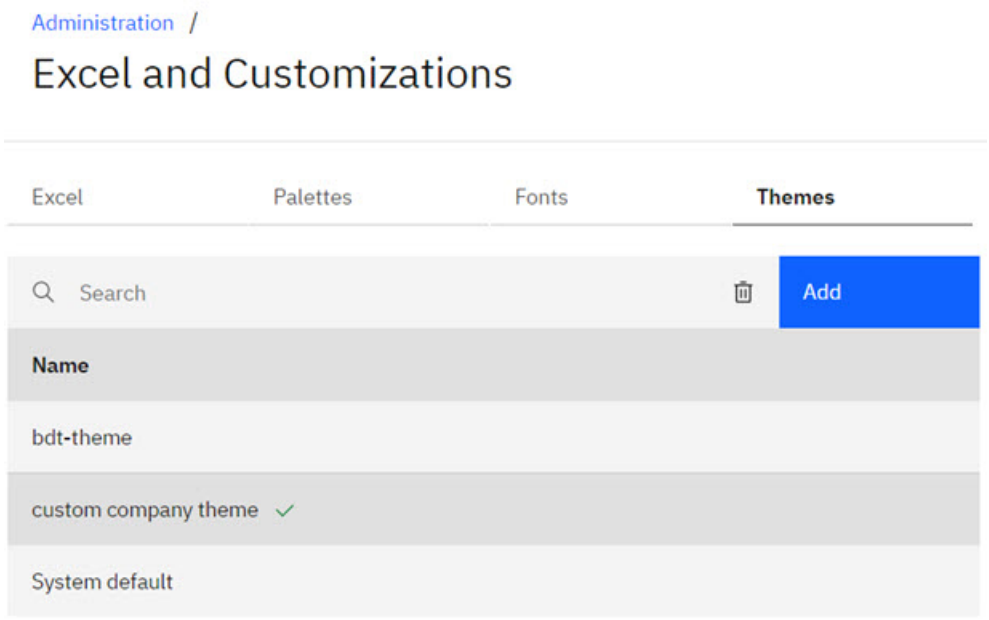
- Planning Analytics Local TM1 Server
- Planning Analytics Workspace
- Planning Analytics for Microsoft Excel
- Planning Analytics Spreadsheet Services/TM1 Web

## Create and apply a custom theme for Planning Analytics Workspace

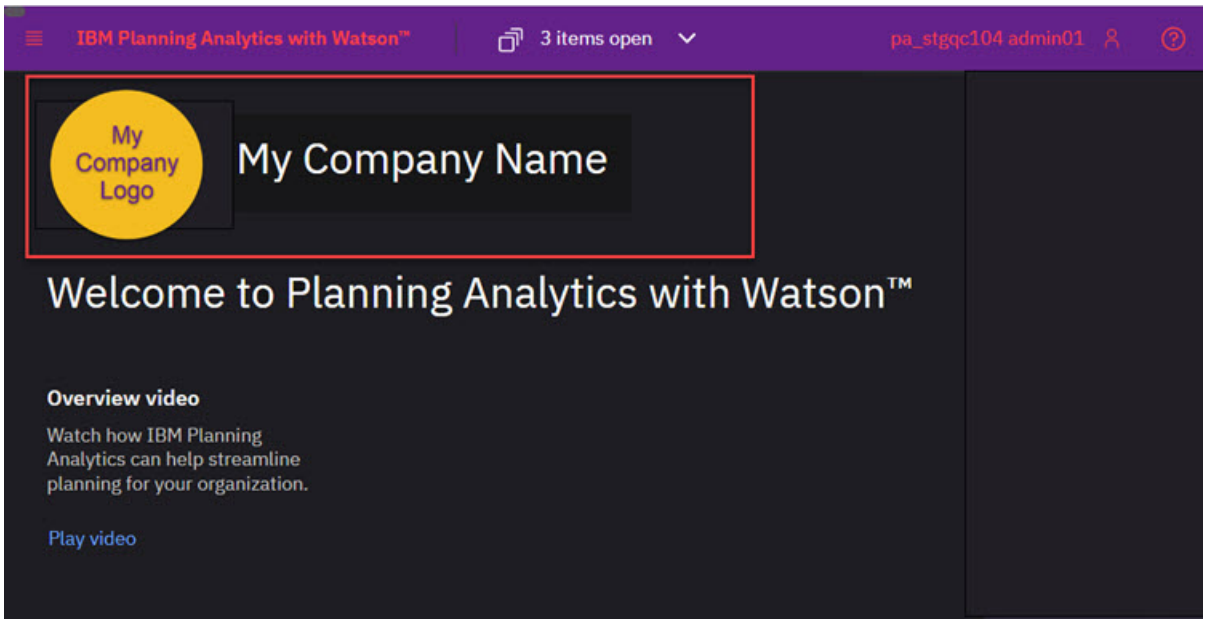
You can now create and upload a custom theme to reflect your corporate branding in Planning Analytics Workspace.

You can use a theme to modify the colors of the application bar and the navigation bar, set the corporate name shown on the home page, and specify an image to display on the home page.

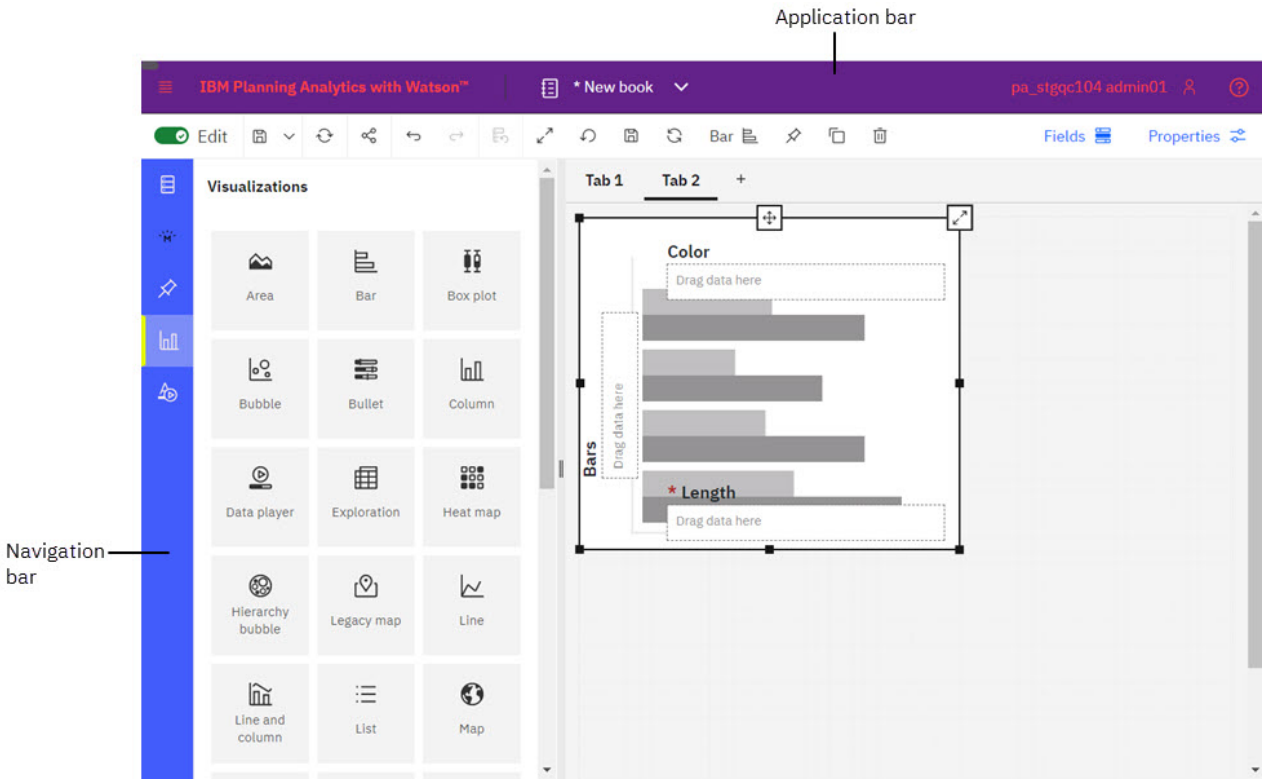
The **Excel and Customizations** page in Planning Analytics Administration on cloud and the **Customizations** page in Planning Analytics Administration local include a new **Themes** tab. You can use the **Themes** tab to upload, manage, and apply themes.




A theme is defined in a `.zip` archive that contains a `.json` file and an `image` directory. The keys in the `.json` specify the colors and images in you interface. For example, you can specify an image and company name on the home page.




Similarly, you can set colors for the application bar and navigation bar.

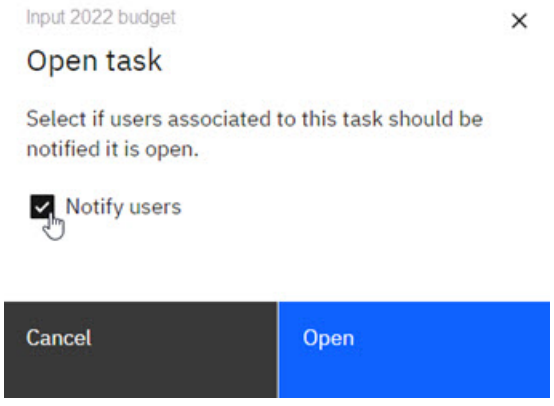



For full details on creating and managing themes, see Customize the interface for your company when Planning Analytics Workspace 2.0.76 is released.

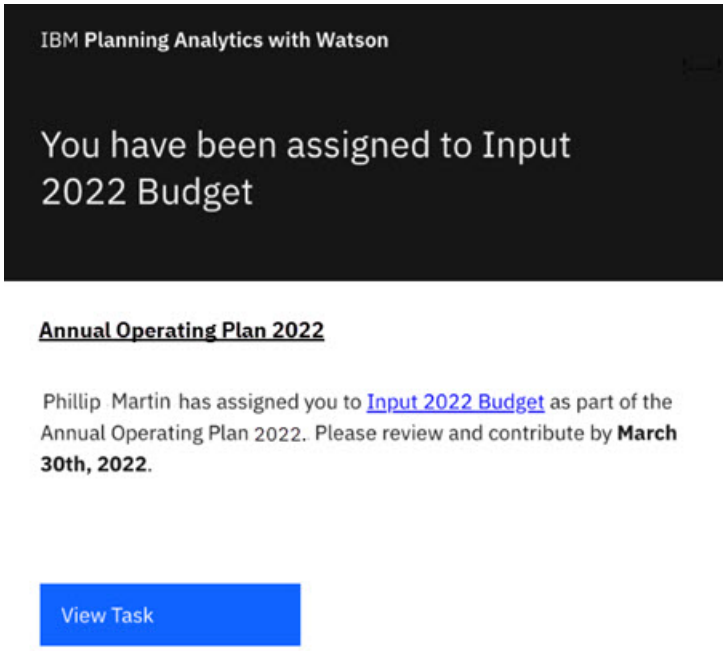
## Send email notification when you open a plan step

You can now send an email notification to contributors when you open a step in a plan.

When a step include at least one asset and has at least one user group assigned to it, you'll receive the option to **Notify users** when you open the step in edit mode.




If you select the **Notify users** option and then click **Open** to open the step, all users in the groups assigned to the step receive an email notification similar to this, including a direct link to the plan step.



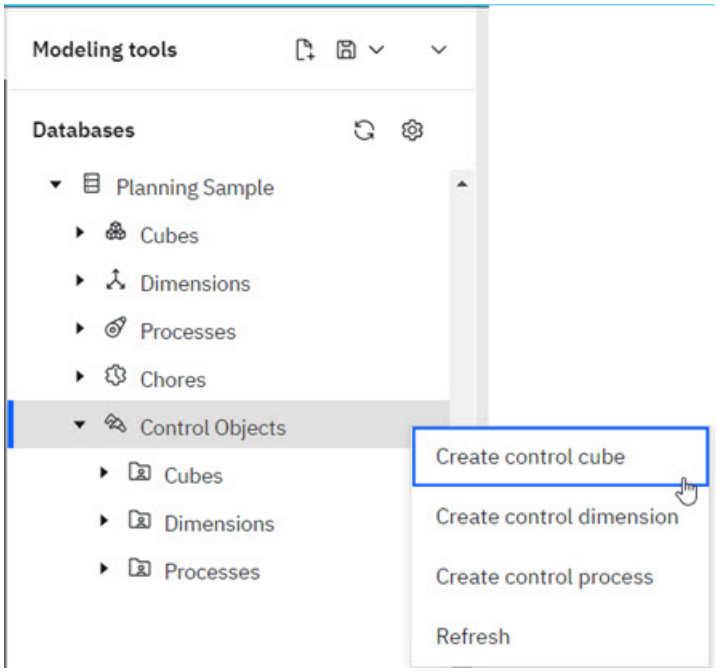

If the Everyone group is assigned to the step, email is **not** sent to members of the Everyone group when the step is opened.

**Note:** The **Notify users** option appears only when you open a step in a plan that is open. If you open a step in a closed plan, there is no option to send email notification to users.

For more information, see Manage a plan.

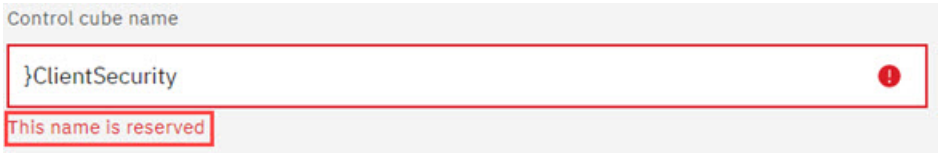
## Create control objects on the modeling workbench

You can now create cube, dimension, and process control objects on the modeling workbench.




To create a control object, click ⋮ next to the **Control Objects** node, then click the appropriate option for the type of control object you want to create.

When you make a selection, you must first enter an object name before you proceed to the appropriate editor. All system control object names are reserved and validated when you create a new control object. If you attempt to create a new control object with a reserved name, you receive a notification.

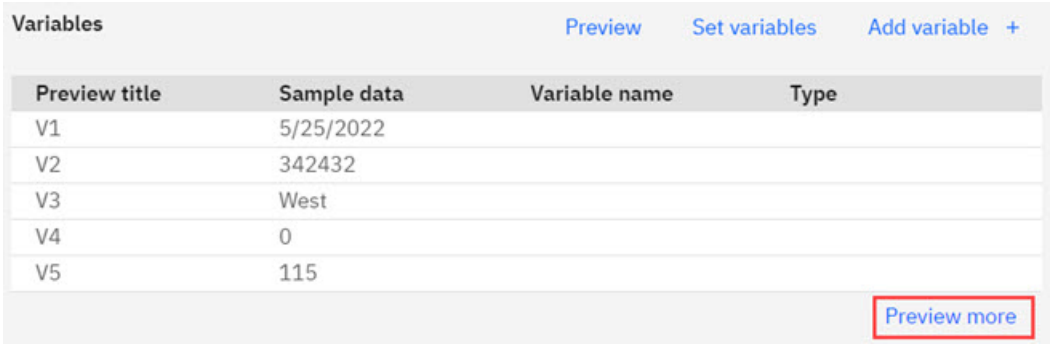



For more information, see Control objects.

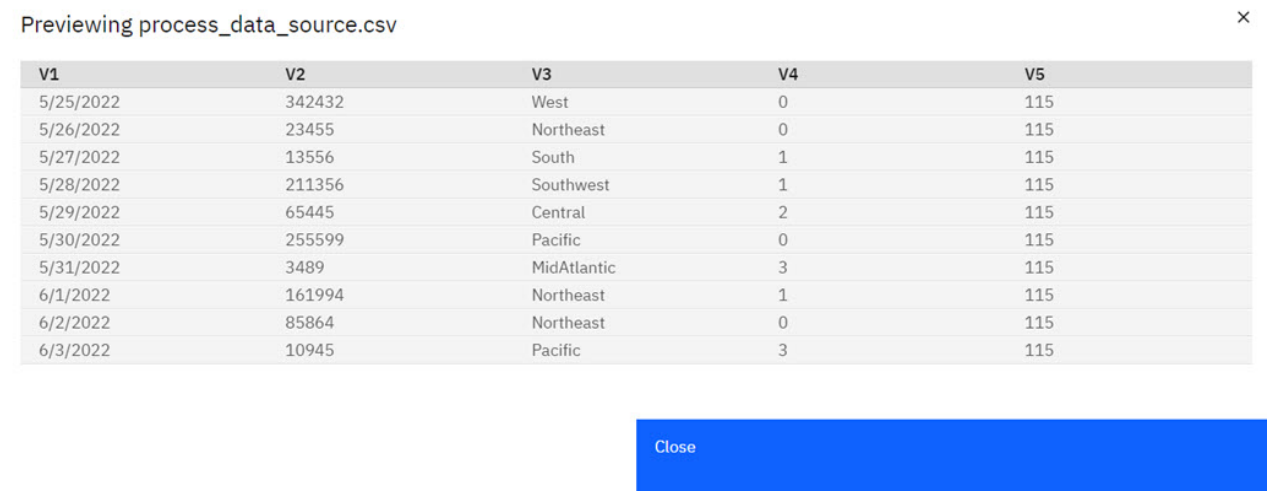
## Preview additional data source rows in the process editor

When you are creating or editing a process with a database connection or file data source, you can now preview additional rows to assist you in managing your process.

After you click **Preview** to initially preview your data source, you'll see a new **Preview more** option.




Click **Preview more** to view the first ten rows in your data source. These additional rows can help you better understand your data source and structure. The additional rows can also help you to confirm that you have loaded the intended data source.

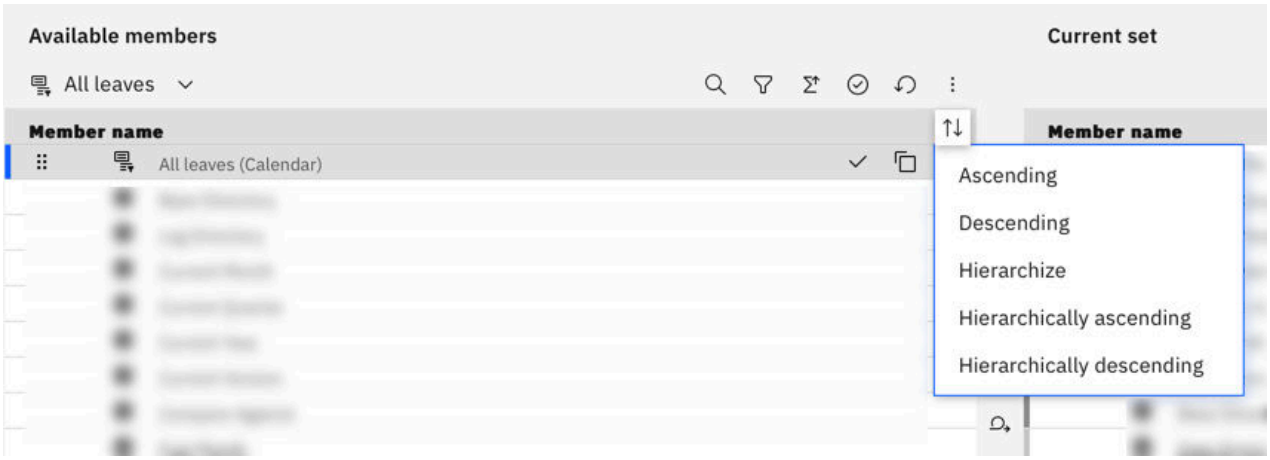



## Sorting available members in the new experience set editor

You can now sort members, attributes, and properties under **Available members** in the new experience set editor. Previously, you could sort only the **Current set**.

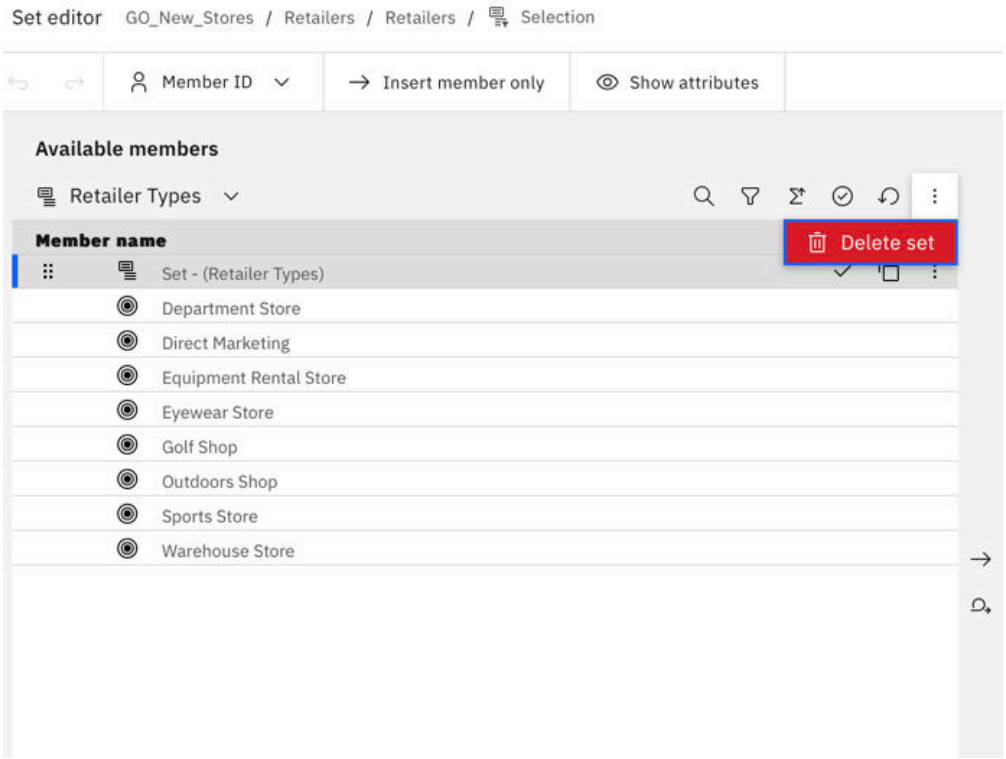
**Note:** To use the new experience view and set editor, you need to enable the feature under **Features** in **Administration**. For more information, see Manage features.

Click **Sort** next to the member, attribute, or property name under **Available members** and select how you want to sort the column. The **Hierarchize** option reverts any sorting you applied to the member's column.




## Delete sets from the new experience set editor

An additional way to delete sets is now available. Previously, you could delete sets only from the navigation tree but now you can also delete sets from the new experience set editor.




**Note:** You cannot restore a set after deleting it.

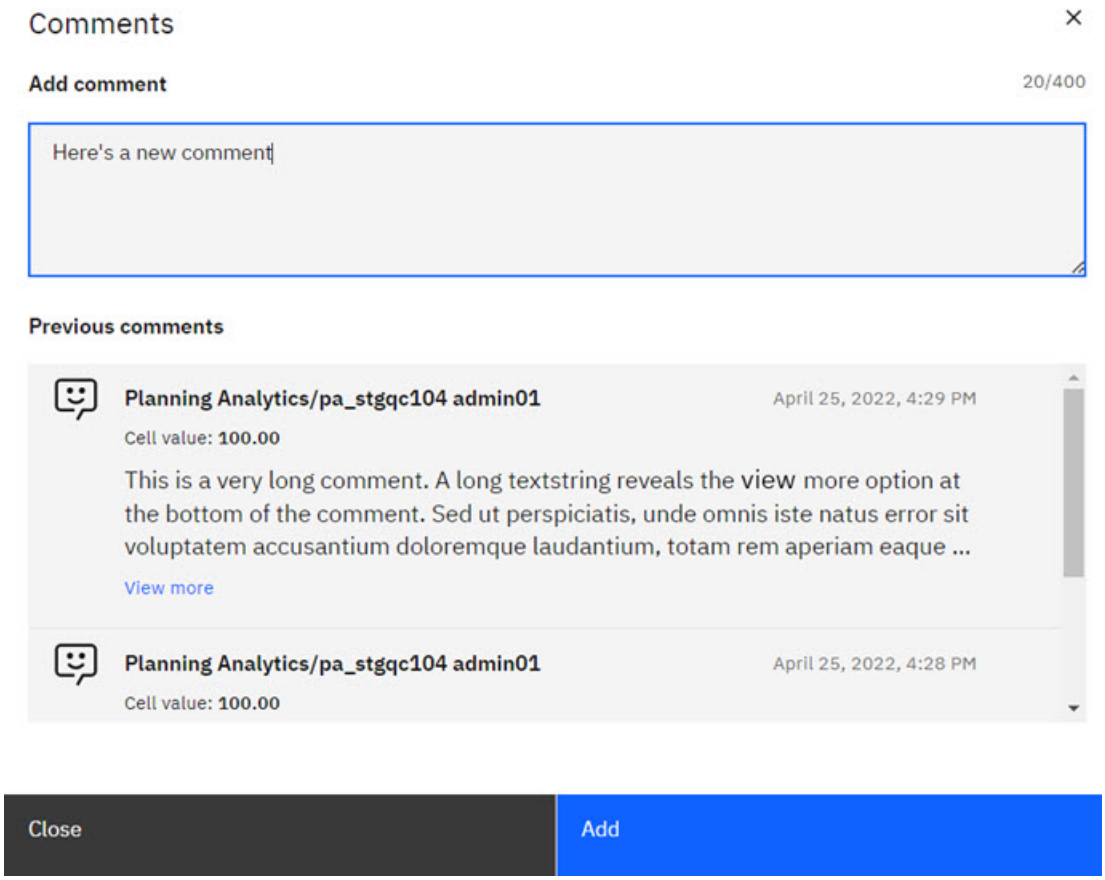
Unnamed sets such as **All roots**, **All members**, and **All leaves** cannot be deleted.

As an administrator, you can delete any public or private set that you own. If you are not an administrator, you can delete only the private sets that you own.

For more information, see Deleting sets.

## Use the updated cell comment feature in the new experience cube viewer

The cell comments feature in the new experience cube viewer has a new look and feel, making it easier to read and enter comments.




Comments are listed in descending order by date/time, with the most recent comment first in the list.

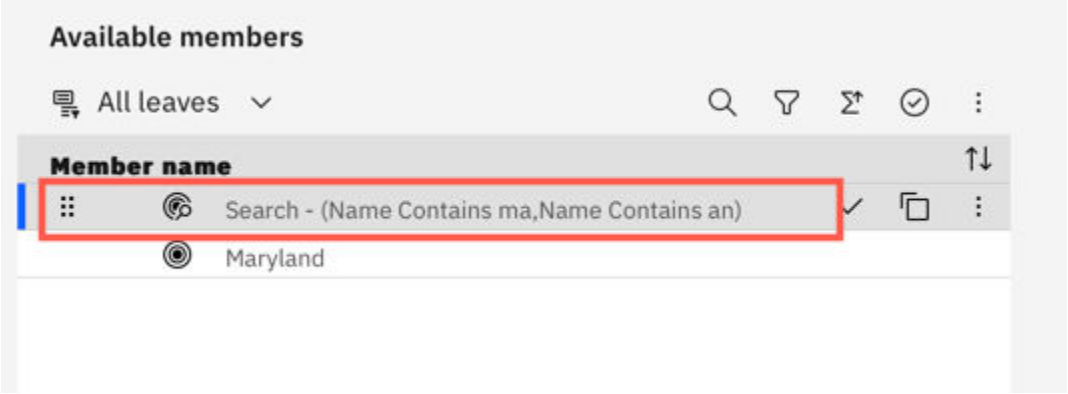
For more information, see Comments.

## Enhanced search and filter features

The search and filter features for the **Available members** pane in the new experience set editor have been enhanced. Before Planning Analytics Workspace 2.0.76, you could use **Search** to search for members in a set and use **Filter** for advanced search options. Now, both these features are more intuitive and improved for usability.

Two new options are now available when you filter a set, **Match all of the following** and **Match any of the following**. These options apply to the filter statements when you filter a set by more than one attribute. The default selection is **Match all of the following**.

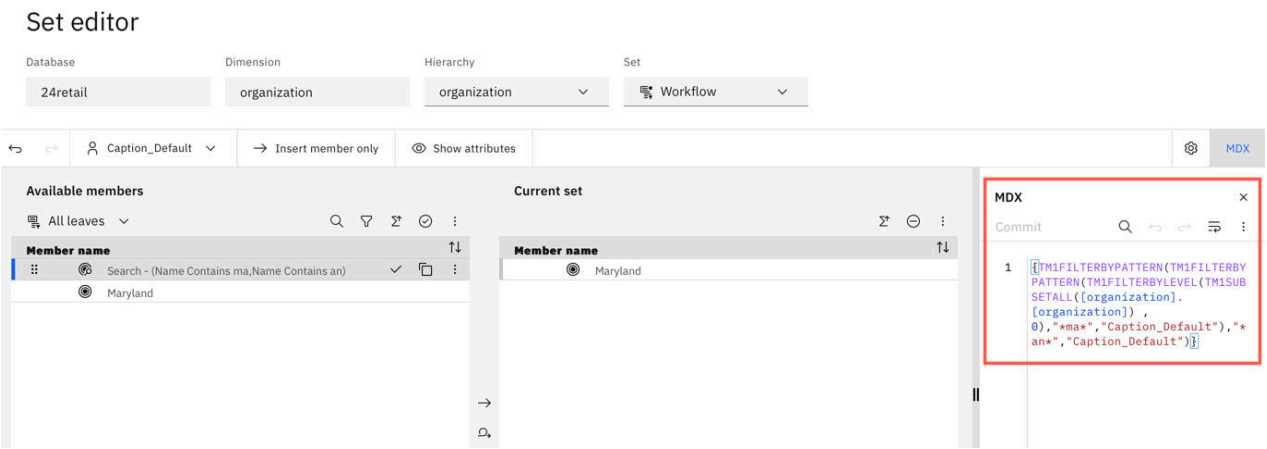
In the following example, the set was filtered to include only members that contain `ma` and `an` in their name.




Once you apply a filter, the filter parameter clears from the **Filter** menu. This allows you to add more filters to further narrow your member selections.

However, you can still see the applied filters in the **Member name** list under **Available members** or in the **MDX**.

**Note:** Replace the **Current set** with the filtered set and click **MDX** to see the applied filters.




If you apply a filter to an already filtered set, the new filter applies to the MDX state around the first filter. For example, in the following MDX, a new filter was applied to an already filtered set to include only members with names that contain the letter `r`.



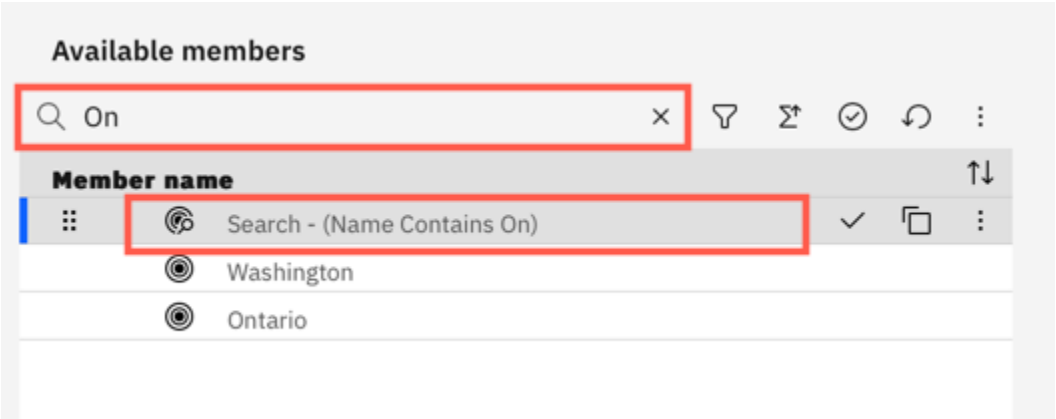

You can click **Reset** to clear all the filters you applied and restore the set to its original state.

For more information on filtering, see Filtering sets.




With **Search**, you can look for members that match specific criteria but only in the current MDX state. This means if a set is filtered, search looks for members that match the given criteria in the filtered set and not the full list of members.

To search for members, click **Search** and type your criteria in the search box. Press Enter to search.




**Note:** Unlike filters, searches do not accumulate. Your search results replace any previous searches that you conducted.

You can clear your search by deleting or replacing the input in the search box and pressing Enter. Or, click **Reset**.

For more information, see Searching in sets.

## New properties for tree map visualizations

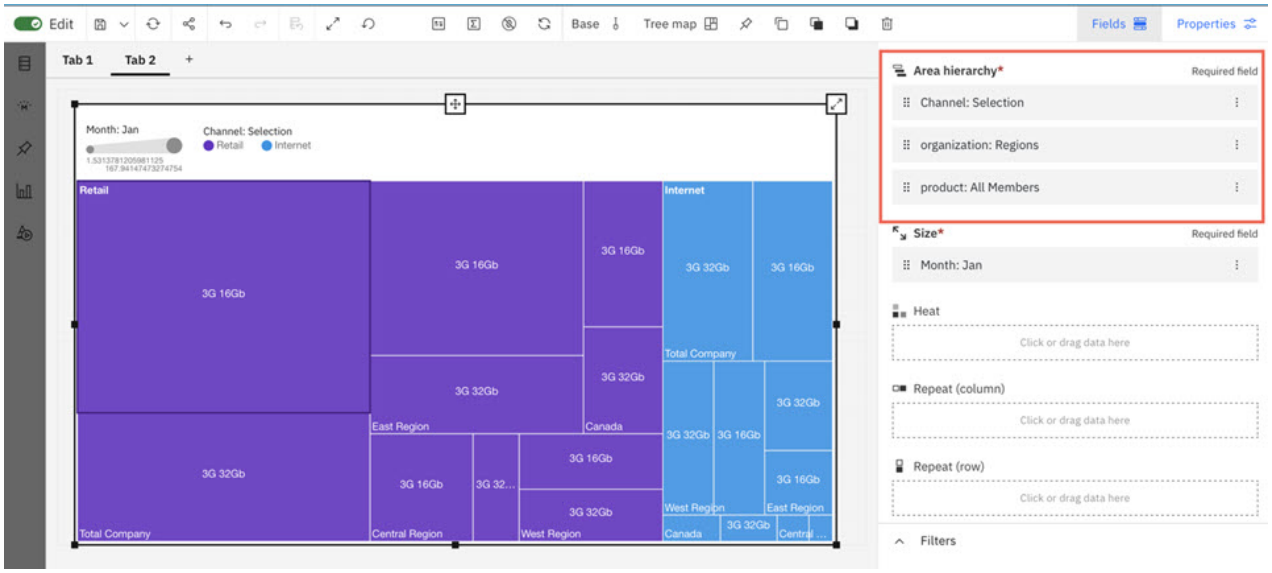
Three new properties, **Show leaf labels**, **Leaf label format**, and **Show intermediate labels** are now available for **Tree map** visualizations. Use these properties to customize the appearance of the visualization.

Enable **Show leaf labels** to display labels in the **Tree map**.

When the **Show leaf labels** property is enabled, you can select the **Leaf label format** to display the item, value of the item, or both in the **Tree map**.

If there are three row dimensions in the row axis of a **Tree map** (in the **Area hierarchy** slot under **Fields**), the labels for intermediate dimension members display by default. An intermediate dimension is the second dimension in a stack of three row dimensions.

In this example, the intermediate dimension is **organization** since it is listed between **channel** and **product**.




Turn off the **Show intermediate labels** property to prevent the labels for intermediate dimension members from displaying in the **Tree map**.

To use these properties:

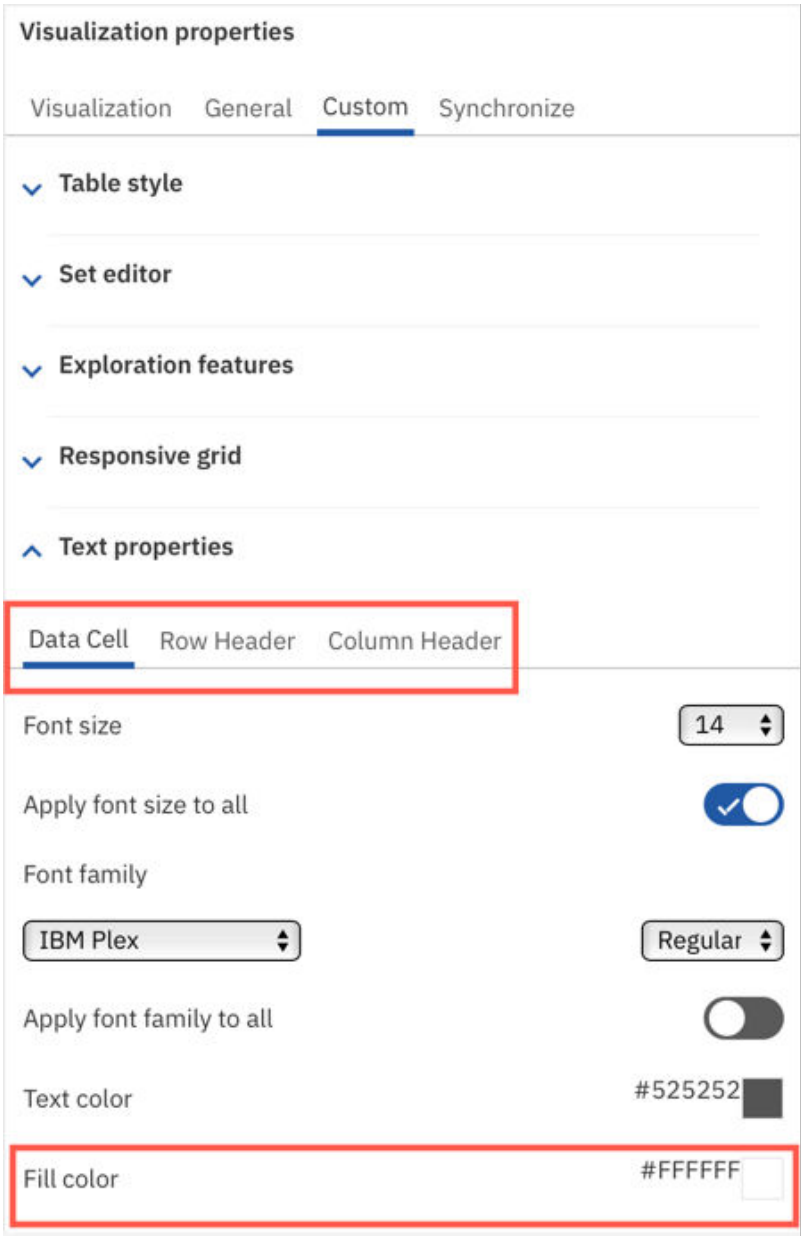
1. Select the **Tree map** in **Edit** mode.
2. Click **Properties**.
3. Under the **Visualization** tab, expand **Chart** to see **Show leaf labels** and, if applicable, **Show intermediate labels**. The **Leaf label format** property displays only if **Show leaf labels** is enabled.

## Customize the background color for data and header cells

In the new experience view, you can now customize the background color of data cells and row and column header cells to any color from the palette or custom color selector.

To customize the background color of cells in the new experience view:

1. In **Edit** mode, go to **Properties**.
2. Click **Custom** and expand **Text properties**.
3. Select the cell type for which you want to change the color.
4. Click **Fill color** and select a new color. Optionally, use **Select custom color** to create your own color with the **Color picker**.



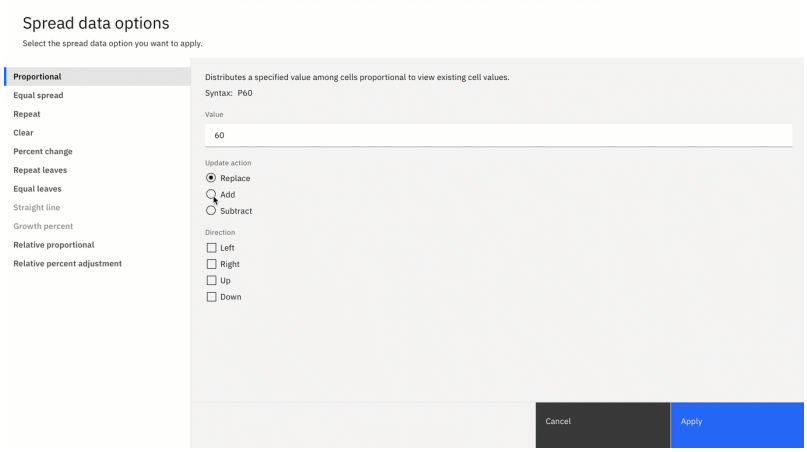

## Updated design for data spread options

As part of the ongoing effort to standardize the Planning Analytics Workspace user experience, the **Spread data options** dialog box in the new experience view has been updated.

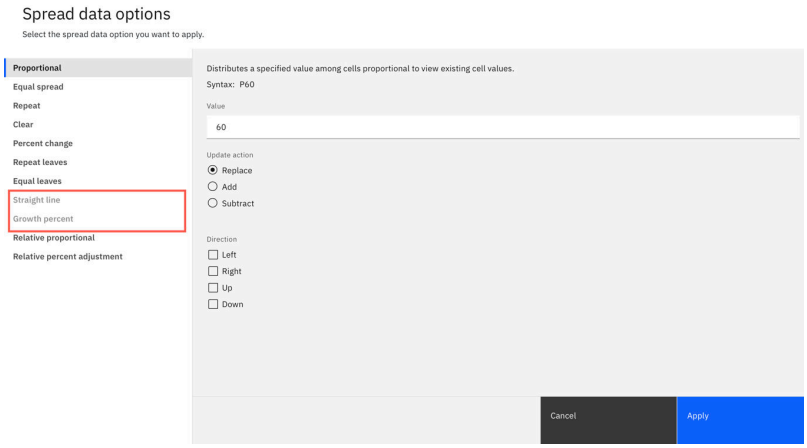
There is no change to the data spreading options or how they function but the new design includes usability improvements.

Each data spread option now includes the **Syntax** value for reference. This is the data spreading syntax that you can enter directly in a cell to accomplish the same spreading that you are specifying in the Spread data options dialog box.

As you select an action or direction, the **Syntax** changes .




Data spreading options and fields that are not valid for the cell or range, display in a dimmed color and are not available for use.

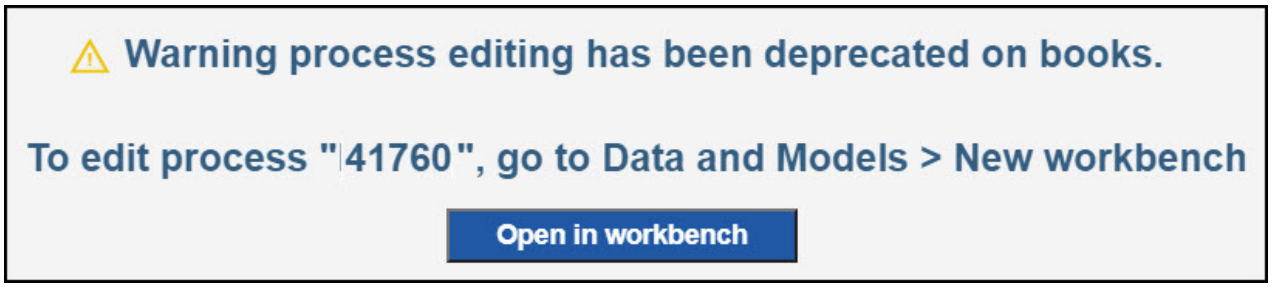



For more information, see Data spreading.

## Deprecation notice - process editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.79

Starting with Planning Analytics Workspace 2.0.79, processes must be created and edited on a modeling workbench.

If you are an administrator or modeler and you attempt to open a process in a book in 2.0.79, you'll receive a notification that process editing must be performed on a workbench. You can click **Open in workbench** to immediately open the process on a new workbench.



## Deprecation notice for IBM Planning Analytics Workspace Classic experience

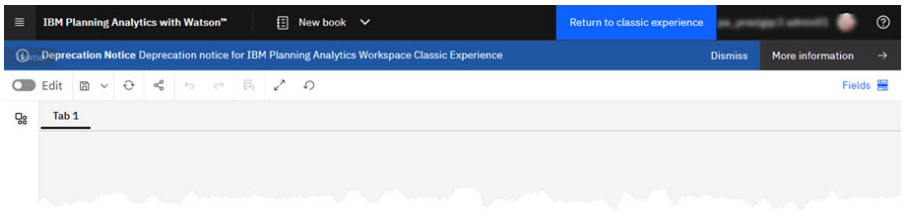
The IBM Planning Analytics team intends to remove the capability to remain on Planning Analytics Workspace Classic and preview the Planning Analytics Workspace new experience as part of the Planning Analytics Workspace 2.0.79 update in August 2022.

The 'new experience' is a major update to the Planning Analytics Workspace user experience released as part of the 2.0.56 update in October 2020. The Planning Analytics Workspace new experience provides an updated look and feel, improved navigation and content management. It also introduces Applications and Plans, a dedicated modeling environment, and Predictive Forecasting capabilities.

The user experience in 2.0.55 and earlier releases is referred to as Planning Analytics Workspace Classic. Cloud environments provisioned before the 2.0.56 update could remain on Planning Analytics Workspace Classic, preview the new experience, or permanently switch to the new experience.

Planning Analytics Workspace environments that use the Classic experience will automatically be updated to the new experience as part of the 2.0.79 update in August 2022. Planning Analytics cloud customers that use the Classic experience are encouraged to preview and switch to the new experience before August 2022.

If you are still using Planning Analytics Workspace Classic prior to the August 2022 update, administrators will see a deprecation notice banner on the home page when using Planning Analytics Workspace Classic or previewing the new experience. An administrator can click the **More information** link to view further details on the deprecation of Planning Analytics Workspace Classic.




The deprecation notice is presented to each administrator upon login. One administrator dismissing the banner dismisses the banner only for that administrator; other administrators can still see the banner.

The banner will reappear with each new Planning Analytics Workspace release until an administrator completes the switch to the new experience.

**Note:** This announcement does not impact Planning Analytics on-premise deployments.

# 2.0.75 - What's new, April 7, 2022

Learn about new features and known issues in version 2.0.75 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

### Timeline of previous Planning Analytics releases

For a comprehensive listing of Planning Analytics component releases by month and year, see Timeline for IBM Planning Analytics Releases. You can view versions and release months for the following components:

- Planning Analytics Local TM1 Server
- Planning Analytics Workspace
- Planning Analytics for Microsoft Excel

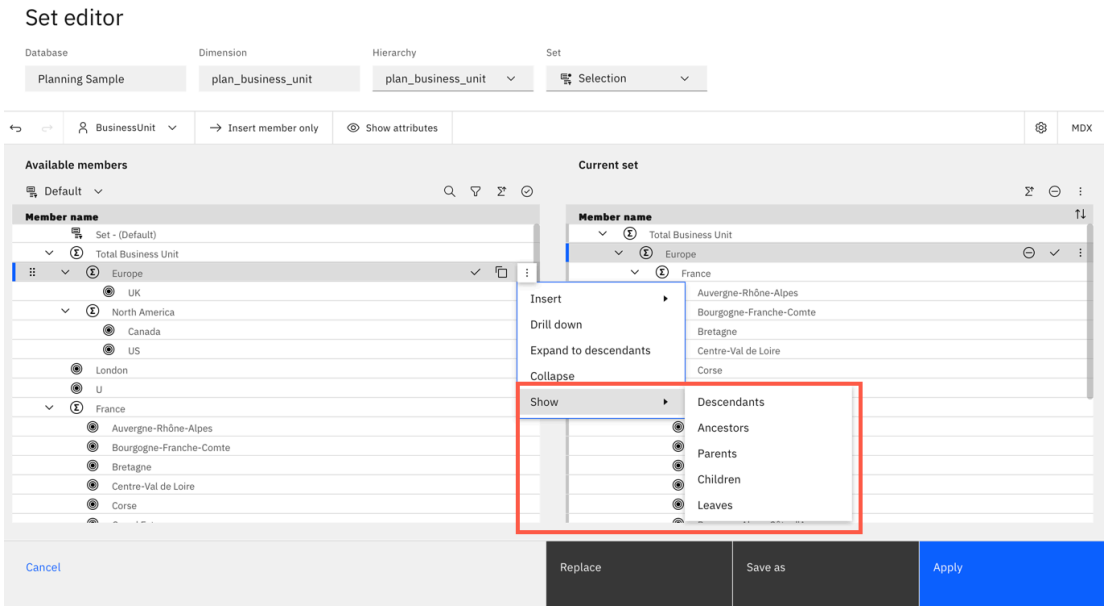
- Planning Analytics Spreadsheet Services/TM1 Web

## Displaying parents, children, or leaves for selected members

The new experience set editor now has three new options in the **Show** menu that allow you to further refine the number of members that you see in a view.

You can now set your view to include only the immediate parents, children, or leaves of selected members.

Select one or more members in the **Available members** or the **Current set** panel. Go to the **Show** menu and choose the related members you want to include in the view. All other members are hidden from the view.




For more information, see Narrowing down selections in the set editor.

## Update to predictive forecasting service

Predictive forecast capabilities in Planning Analytics Workspace use a common library of capabilities based on IBM Watson®. In this Planning Analytics Workspace release, this library has been updated to support additional capabilities such as outlier detection and correction.

This library update should not significantly affect the accuracy of previous forecast results. However, you may notice changes in the forecasted values themselves. The new library creates a more rigorous distinction between training and testing in the historical data, so in some cases, your forecast accuracy may benefit from more historical data.

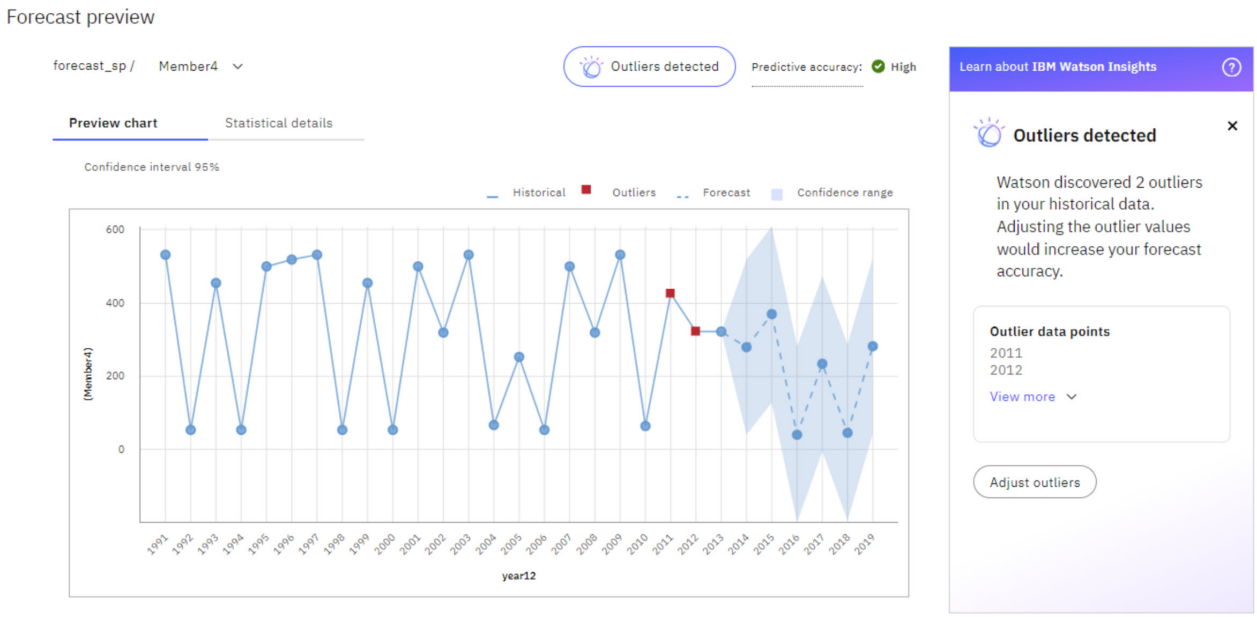
**Note:** In extreme cases, a high number of multiple users simultaneously requesting forecast results might result in a small decrease in performance. This scenario is unlikely, but if performance decreases the 'spinning waiting' icon is displayed until the forecast process is complete.

## Adjust for outliers in your forecast

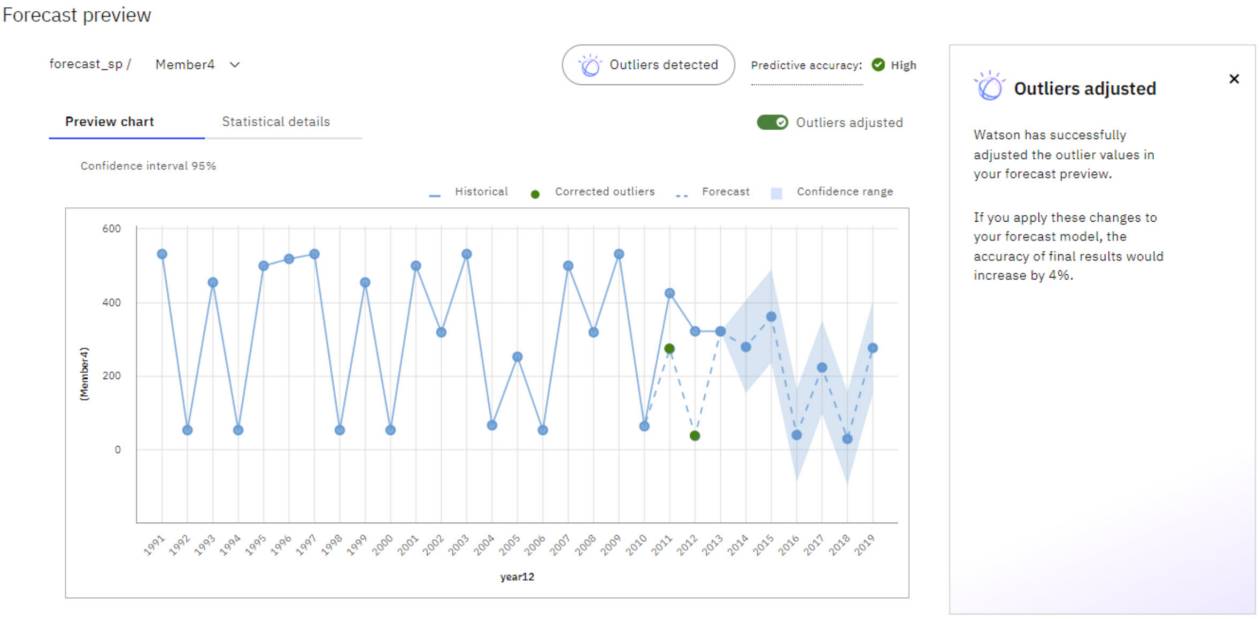
If outliers are detected in your forecast, you can now use an option to adjust a forecast preview to compensate for the outliers and improve the accuracy of your forecast.

Note that you can adjust outliers only after you create a preview.

If outliers are detected in your preview, you receive notification.




Click **Adjust outliers** to update the forecast and see the impact of the adjustment.




An informational message shows the impact of the adjustment. In this example, the adjustment results in a 71% increase in the accuracy of the forecast. You can toggle the **Outliers adjusted** option  to see the forecast with or without the adjustment. If you choose not to accept the adjustment, you can turn off the **Outliers adjusted** option and your forecast will not include the automatic outlier adjustment.

When multiple outliers are detected, they are adjusted sequentially from the beginning of the data set. This means that each adjusted outlier value is applied to the calculation of the adjustment for the subsequent outlier, resulting in a more accurate forecast.

## Changes to cube creation

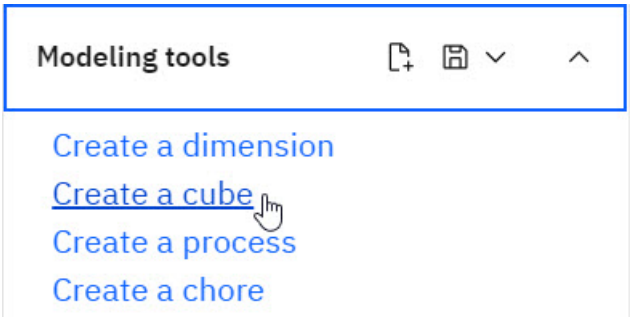
As of Planning Analytics Workspace 2.0.75, you can no longer create a cube from the Data tree in a book.

Cubes can be created only in a Modeling workbench. Additionally, the cube creation process has been modified and an option to create a new cube based on the structure of an existing cube is now available.

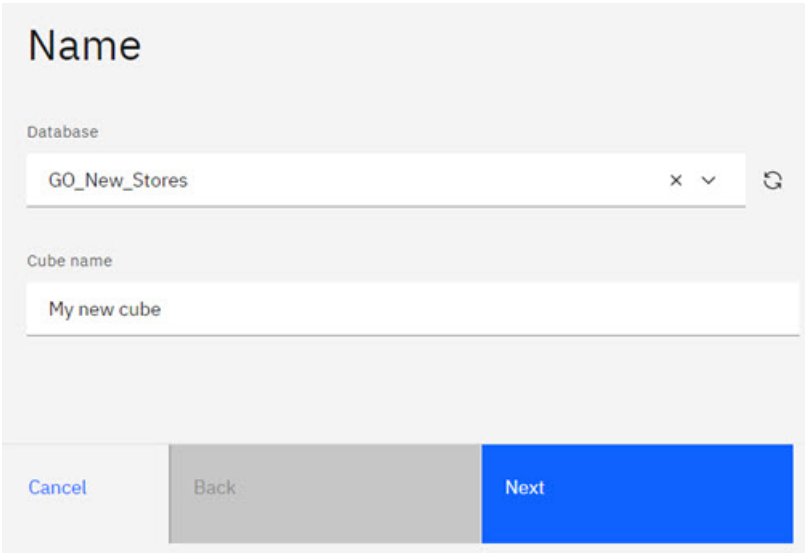
## Create a new cube on the Modeling workbench

To create a cube:

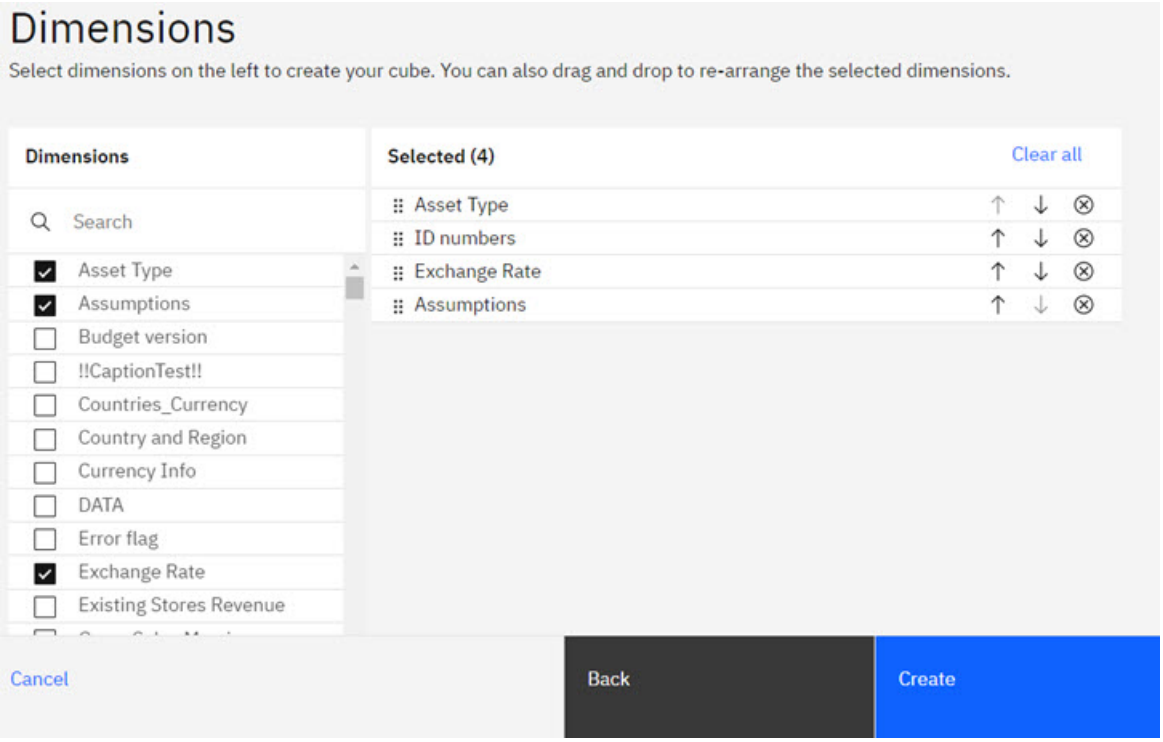
1. Open a Modeling workbench.
2. Expand the **Modeling tools** menu, then click **Create cube**.




3. Select the database on which you want to create the cube.
4. Enter a name for the cube.




5. Click **Next**.
6. Select the dimensions you want to include in the cube from the **Dimensions** list. When you select a dimension, it is added to the **Selected** list.




7. Optionally, you can change the order of dimensions in the cube by clicking a dimension handle and dragging it to a new position or by clicking the **Up** or **Down** arrow next to a dimension.
8. Click **Create** to create the new cube.

### Create a new cube based on the structure of an existing cube

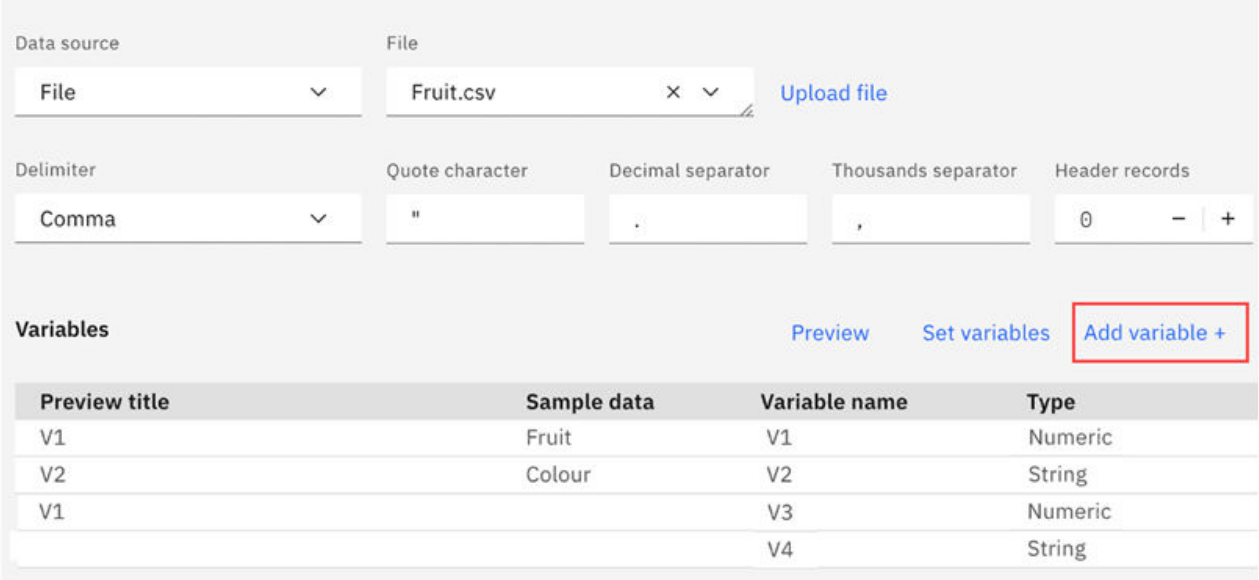
If a cube exists with the structure and dimensionality that you want you use in a new cube, the **Create from cube** option provides a handy shortcut for cube creation.

1. Open a Modeling workbench.
2. In the **Databases** tree, locate the existing cube that you want to use as the basis for a new cube.
3. Right-click the existing cube, then click **Create from cube**.
4. Enter a new **Cube name**, then click **Next**. The dimensions from the existing cube are preselected and ordered as specified in the existing cube.
5. Optionally, you can select additional dimension to include in the new cube, remove preselected dimension, or change the order of dimensions for the new cube.
6. Click **Create**.

## Add variables in the Process editor

Up until now, the number of variables in a Planning Analytics Workspace TurboIntegrator process was limited to the number of columns in the data source. The variable for the process were created automatically, one for each column in the source.

As of Planning Analytics Workspace 2.0.75, you can use the new **Add variable** option on the process editor to add variables to a process.




A variable name cannot be empty and must adhere to the Planning Analytics naming conventions.

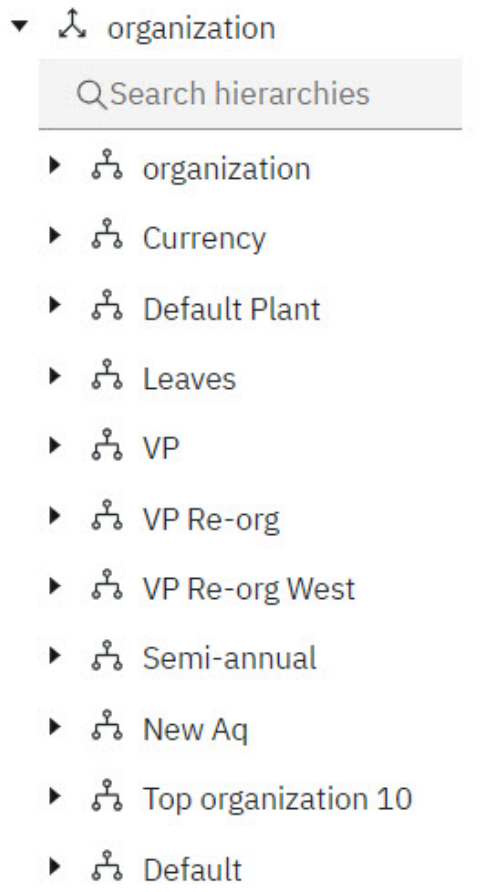
You can also delete any variable in a process. Hover the cursor on any variable name to reveal, and then click, the icon.

The ability to add variables to a process is useful when you are creating a process, but the data source is not available. It's also useful when the number of columns in the data source changes, as can occur when the data source of a process is modified at run time.

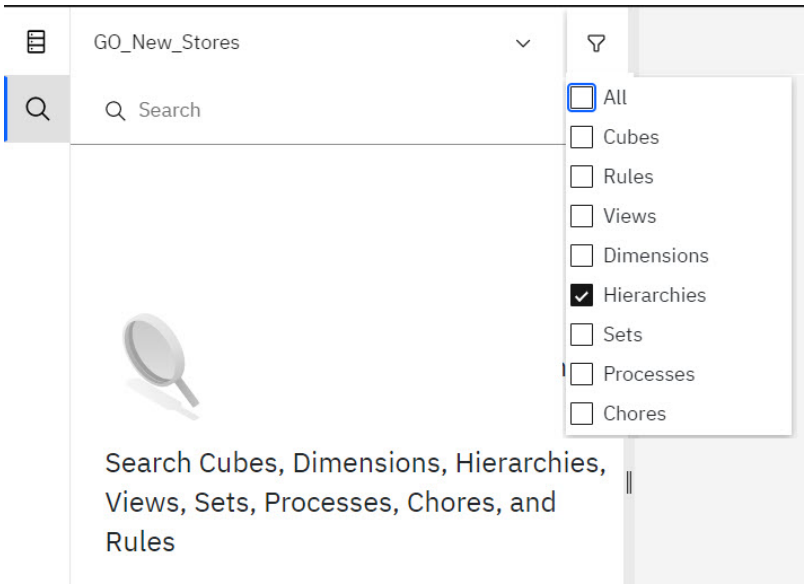
If you have more variables in a process than columns in the data source, the process still works. The extra variables will simply be empty.

## Search and filter on hierarchies

The **Data** tree for books and the **Databases tree** for workbenches now displays a search box under a dimension when the dimension contains 11 or more hierarchies.



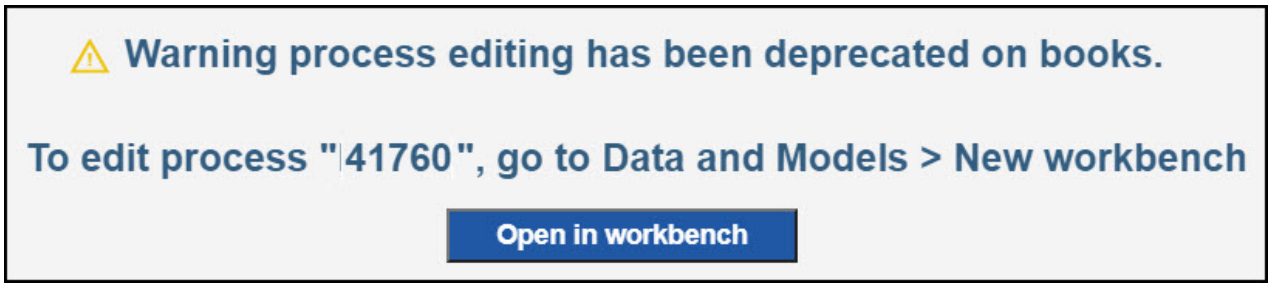

Additionally, the database **Search** on the modeling workbench now encompasses hierarchies, and you can filter search results to include or exclude hierarchies.



## Deprecation notice - process editor to be removed from books, available only on workbenches as of Planning Analytics Workspace 2.0.79

Starting with Planning Analytics Workspace 2.0.79, processes must be created and edited on a modeling workbench.

If you are an administrator or modeler and you attempt to open a process in a book in 2.0.79, you'll receive a notification that process editing must be performed on a workbench. You can click **Open in workbench** to immediately open the process on a new workbench.




## Deprecation notice for IBM Planning Analytics Workspace Classic experience

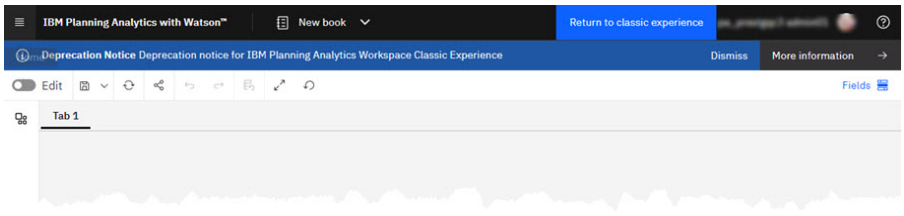
The IBM Planning Analytics team intends to remove the capability to remain on Planning Analytics Workspace Classic and preview the Planning Analytics Workspace new experience as part of the Planning Analytics Workspace 2.0.79 update in August 2022.

The 'new experience' is a major update to the Planning Analytics Workspace user experience released as part of the 2.0.56 update in October 2020. The Planning Analytics Workspace new experience provides an updated look and feel, improved navigation and content management. It also introduces Applications and Plans, a dedicated modeling environment, and Predictive Forecasting capabilities.

The user experience in 2.0.55 and earlier releases is referred to as Planning Analytics Workspace Classic. Cloud environments provisioned before the 2.0.56 update could remain on Planning Analytics Workspace Classic, preview the new experience, or permanently switch to the new experience.

Planning Analytics Workspace environments that use the Classic experience will automatically be updated to the new experience as part of the 2.0.79 update in August 2022. Planning Analytics cloud customers that use the Classic experience are encouraged to preview and switch to the new experience before August 2022.

If you are still using Planning Analytics Workspace Classic prior to the August 2022 update, administrators will see a deprecation notice banner on the home page when using Planning Analytics Workspace Classic or previewing the new experience. An administrator can click the **More information** link to view further details on the deprecation of Planning Analytics Workspace Classic.




The deprecation notice is presented to each administrator upon login. One administrator dismissing the banner dismisses the banner only for that administrator; other administrators can still see the banner.

The banner will reappear with each new Planning Analytics Workspace release until an administrator completes the switch to the new experience.

**Note:** This announcement does not impact Planning Analytics on-premise deployments.

# 2.0.74 - What's new, March 10, 2022

Learn about new features and known issues in version 2.0.74 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

### Timeline of previous Planning Analytics releases

For a comprehensive listing of Planning Analytics component releases by month and year, see Timeline for IBM Planning Analytics Releases. You can view versions and release months for the following components:

- Planning Analytics Local TM1 Server
- Planning Analytics Workspace
- Planning Analytics for Microsoft Excel
- Planning Analytics Spreadsheet Services/TM1 Web

## Windows Server 2019 support (local only)

IBM Planning Analytics Workspace Local versions 2.0.74 and later now support Windows Server 2019. You can install Planning Analytics Workspace Local on Windows Server 2019.

For more information, see Install Planning Analytics Workspace Local on Windows Server 2019.

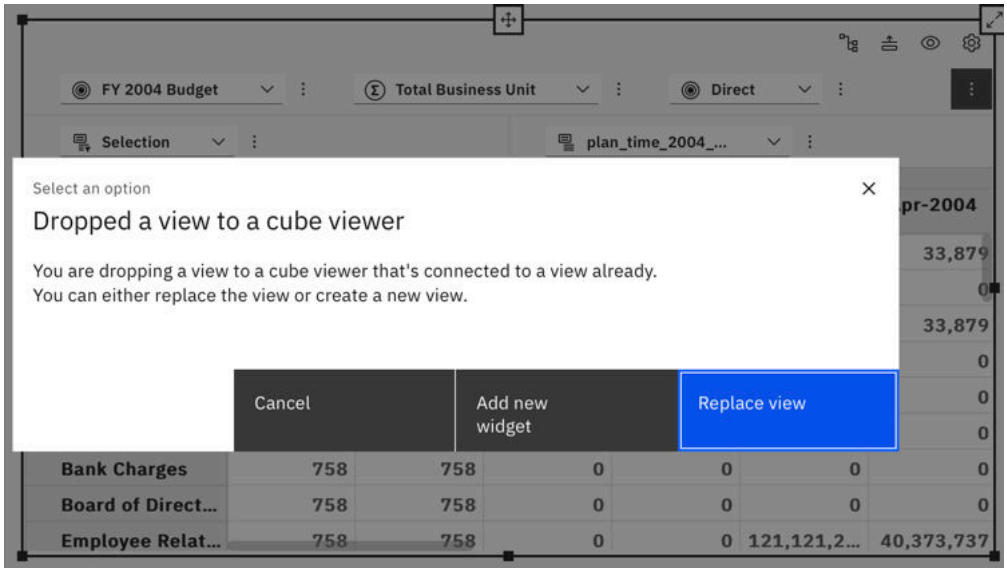
Support for Windows Server 2016 continues as well.

## Add a view by dragging it from the data tree

You can now add a view to a book by dragging the view from the data tree and dropping it on to a cube viewer.

In the data tree, expand the cube that contains the data you want to use. Select the view that you want to add and drag it on to a cube viewer (new or classic).

If you drag a view on a cube viewer that is not empty, you can either replace the existing view with the new one, or add the view as a new widget.

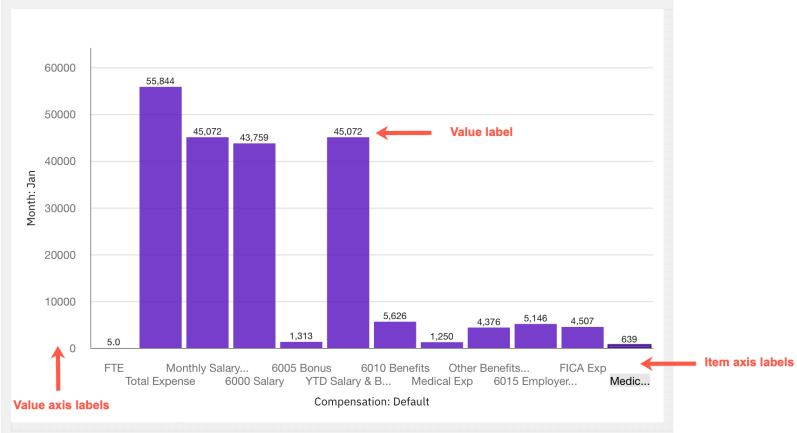


**Note:** When you replace an existing view with a new view, the new view inherits the existing view's properties (such as background color).

## Apply text styles to axis values in visualizations

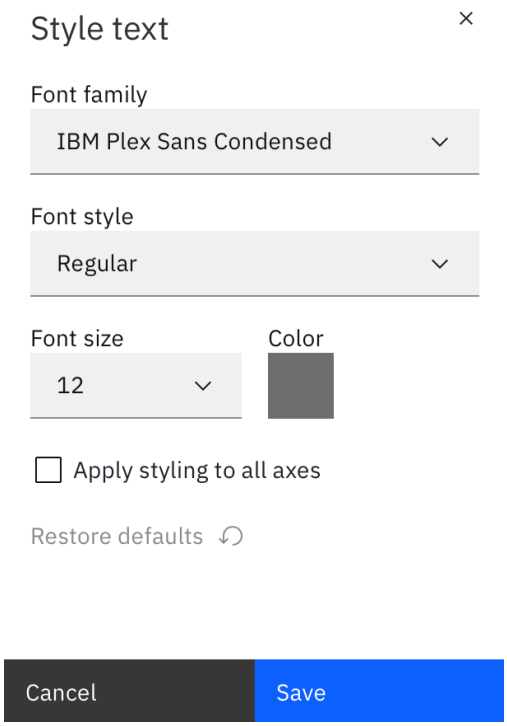
In visualizations, you can now apply text styles to the value axis labels and value labels. Previously, you could apply styling to only item axis labels.

Right-click any axis label or value label and click **Style text** to select and apply text styles.

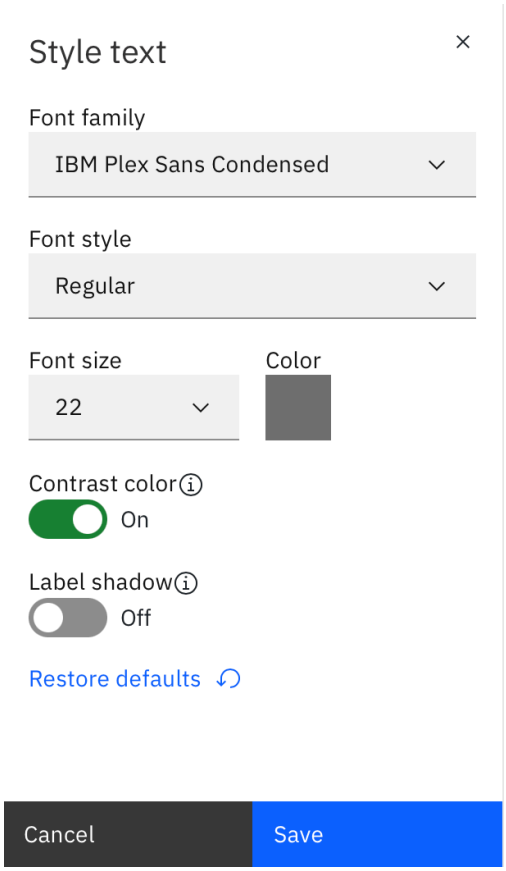



The **Style text** menu includes a new **Font style** option.

For axes, the **Style text** menu includes an option to apply styling to all axes in the visualization.

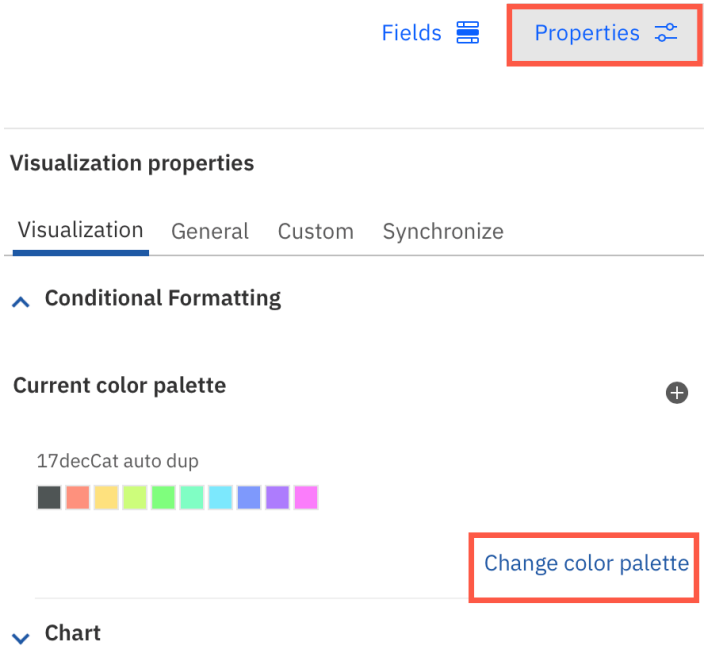



For value labels, the **Style text** menu is slightly different and includes **Contrast color** and **Label shadow** options for accessibility. When **Contrast color** is on, it auto adjusts the font color and overrides the selected color. Turning on **Label shadow** adds a shadow to value labels.




## Availability of color palettes for conditional formatting in Properties

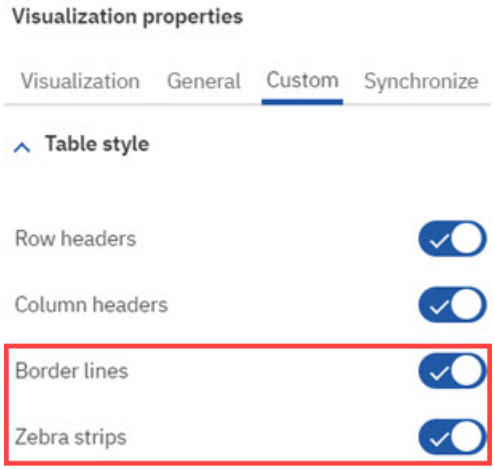
Global and custom color palettes are now available for views with conditional formatting and can be applied from **Properties**.



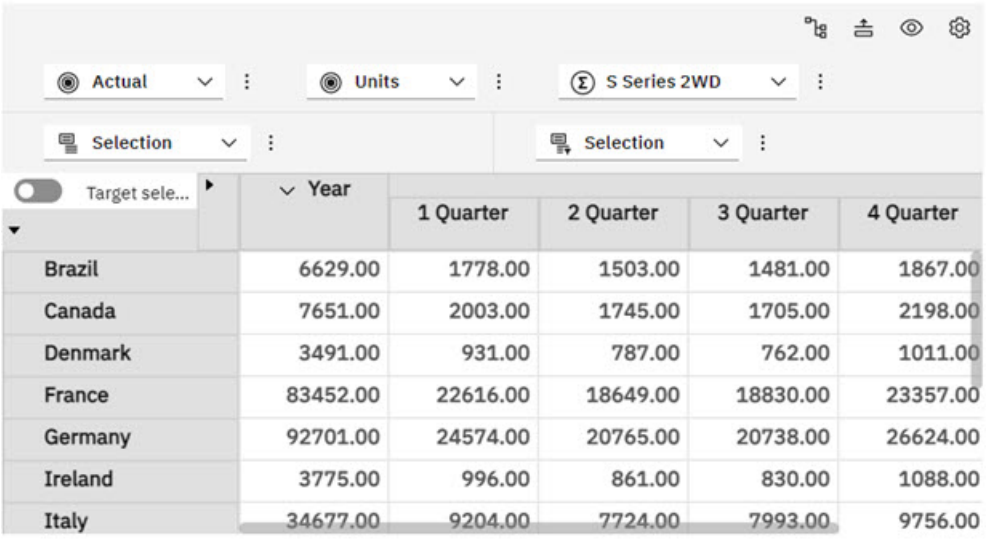

Previously, you could only use default colors when applying conditional formatting to views. You can change these default colors now by clicking **Change color palette** under **Visualization** in **Properties**. For more information, see Conditional formats.

## Customize your explorations with new properties

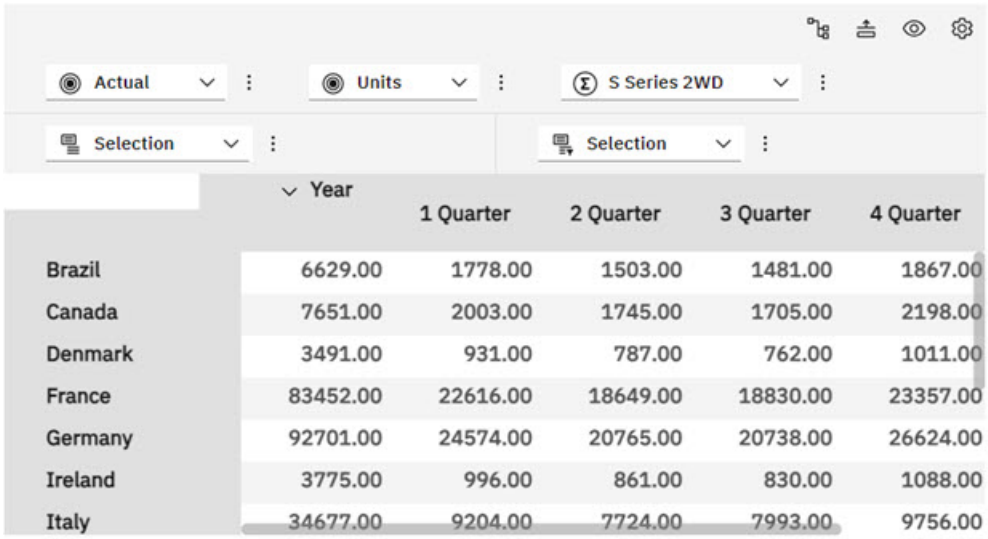
Two new properties, **Border lines** and **Zebra strips**, are now available to let you customize the appearance of new experience explorations (views). Both of these properties were previously applied with themes in classic explorations.




The **Border lines** property controls the display of border lines around cells and header members in an exploration. Enable the property to display border lines, disable it to hide border lines. This example shows an exploration with the **Border lines** property enabled.




The **Zebra strips** property controls row shading. Enable the property to apply a shaded background to alternating rows in an exploration, disable it to display all rows with a uniform background. This example shows an exploration with the **Zebra strips** property enabled.




To use these properties:

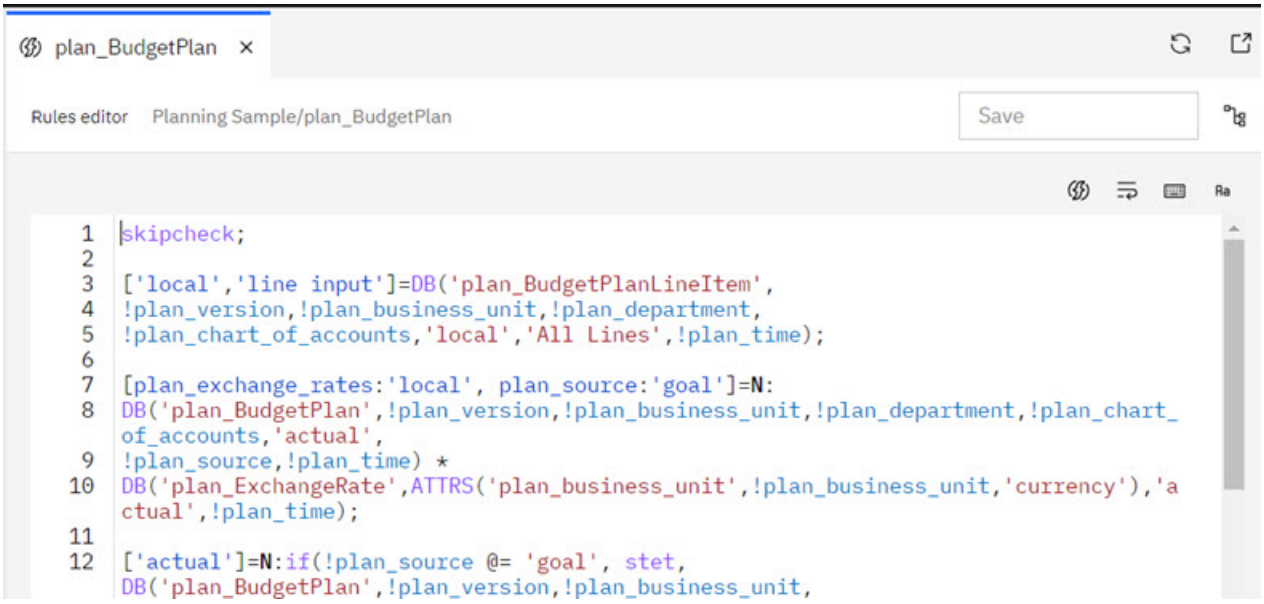
1. Select an exploration while in Edit mode.
2. Click **Properties**.
3. Click the **Custom** tab.
4. Expand the **Table style** group to reveal the **Border lines** and **Zebra strips** properties.

The new experience exploration was introduced in Planning Analytics Workspace 2.0.71. You can review the features and capabilities of the new experience exploration here.

## Updated rules editor

As part of the ongoing effort to standardize the Planning Analytics Workspace user experience, the rules editor has been updated.

Note that any existing rules editors that were previously saved in a modeling book or workbench remain available using the previous interface in the modeling book or workbench.




For more information, see Use the rules editor in Planning Analytics Workspace.

## Deprecation notice for IBM Planning Analytics Workspace Classic experience

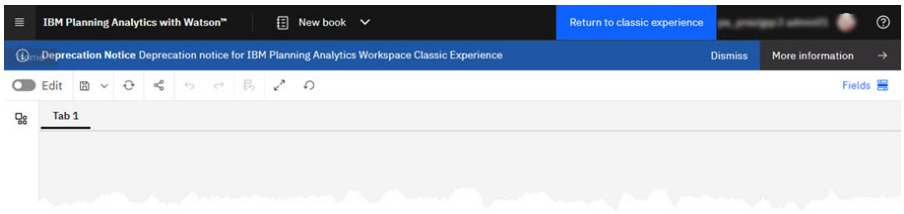
The IBM Planning Analytics team intends to remove the capability to remain on Planning Analytics Workspace Classic and preview the Planning Analytics Workspace new experience as part of the Planning Analytics Workspace 2.0.79 update in August 2022.

The 'new experience' is a major update to the Planning Analytics Workspace user experience released as part of the 2.0.56 update in October 2020. The Planning Analytics Workspace new experience provides an updated look and feel, improved navigation and content management. It also introduces Applications and Plans, a dedicated modeling environment, and Predictive Forecasting capabilities.

The user experience in 2.0.55 and earlier releases is referred to as Planning Analytics Workspace Classic. Cloud environments provisioned before the 2.0.56 update could remain on Planning Analytics Workspace Classic, preview the new experience, or permanently switch to the new experience.

Planning Analytics Workspace environments that use the Classic experience will automatically be updated to the new experience as part of the 2.0.79 update in August 2022. Planning Analytics cloud customers that use the Classic experience are encouraged to preview and switch to the new experience before August 2022.

If you are still using Planning Analytics Workspace Classic prior to the August 2022 update, administrators will see a deprecation notice banner on the home page when using Planning Analytics Workspace Classic or previewing the new experience. An administrator can click the **More information** link to view further details on the deprecation of Planning Analytics Workspace Classic.




The deprecation notice is presented to each administrator upon login. One administrator dismissing the banner dismisses the banner only for that administrator; other administrators can still see the banner.

The banner will reappear with each new Planning Analytics Workspace release until an administrator completes the switch to the new experience.

**Note:** This announcement does not impact Planning Analytics on-premise deployments.

# 2.0.73 - What's new, February 10, 2022

Learn about new features and known issues in version 2.0.73 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

### Timeline of previous Planning Analytics releases

For a comprehensive listing of Planning Analytics component releases by month and year, see Timeline for IBM Planning Analytics Releases. You can view versions and release months for the following components:

- Planning Analytics Local TM1 Server
- Planning Analytics Workspace
- Planning Analytics for Microsoft Excel

- Planning Analytics Spreadsheet Services/TM1 Web

## Deprecation notice for IBM Planning Analytics Workspace Classic experience

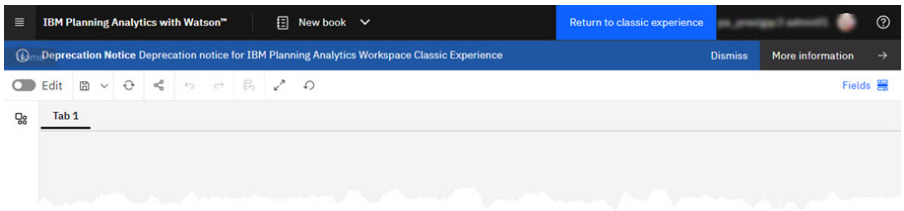
The IBM Planning Analytics team intends to remove the capability to remain on Planning Analytics Workspace Classic and preview the Planning Analytics Workspace new experience as part of the Planning Analytics Workspace 2.0.79 update in August 2022.

The 'new experience' is a major update to the Planning Analytics Workspace user experience released as part of the 2.0.56 update in October 2020. The Planning Analytics Workspace new experience provides an updated look and feel, improved navigation and content management. It also introduces Applications and Plans, a dedicated modeling environment, and Predictive Forecasting capabilities.

The user experience in 2.0.55 and earlier releases is referred to as Planning Analytics Workspace Classic. Cloud environments provisioned before the 2.0.56 update could remain on Planning Analytics Workspace Classic, preview the new experience, or permanently switch to the new experience.

Planning Analytics Workspace environments that use the Classic experience will automatically be updated to the new experience as part of the 2.0.79 update in August 2022. Planning Analytics cloud customers that use the Classic experience are encouraged to preview and switch to the new experience before August 2022.

If you are still using Planning Analytics Workspace Classic prior to the August 2022 update, administrators will see a deprecation notice banner on the home page when using Planning Analytics Workspace Classic or previewing the new experience. An administrator can click the **More information** link to view further details on the deprecation of Planning Analytics Workspace Classic.




The deprecation notice is presented to each administrator upon login. One administrator dismissing the banner dismisses the banner only for that administrator; other administrators can still see the banner.

The banner will reappear with each new Planning Analytics Workspace release until an administrator completes the switch to the new experience.

**Note:** This announcement does not impact Planning Analytics on-premise deployments.

## Creation of modeling books no longer supported

As previously announced in a deprecation notice, the ability to create modeling books is no longer available in Planning Analytics Workspace as of 2.0.73. Existing modeling books continue to be usable, but all modeling activities are now supported in the Modeling Workbench.

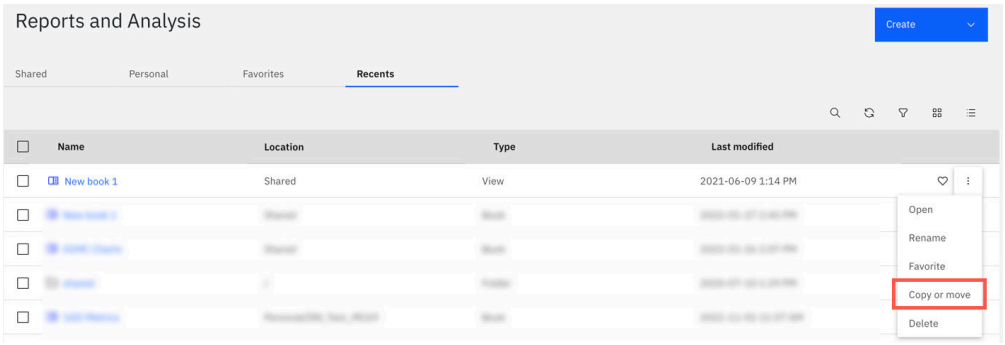
You can review the deprecation notice here.

For more information on the Modeling Workbench, see Use the Modeling Workbench to create and manage your model.

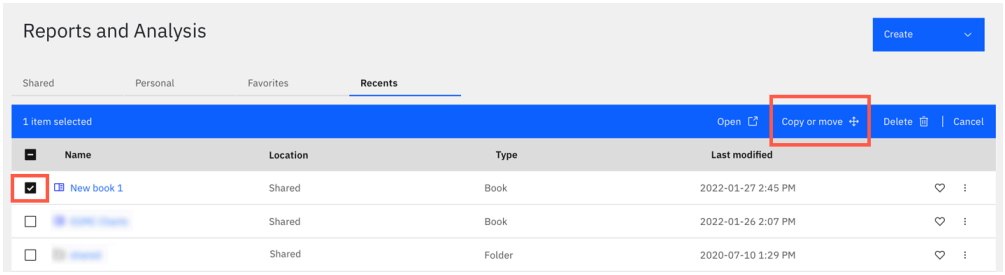
## Copy assets or folders to a shared or personal folder

All assets and folders in your **Shared** and **Personal** folders can now be copied to a new location. A new **Copy or move** option lets you copy (or move) an asset or folder to a new location, either to or within the **Shared** or **Personal** folder.

Click the menu icon next to any asset or folder to see the **Copy or move** option.




You can also select the asset or folder to see a new **Copy or move** option in the toolbar.




Click either of the two **Copy or move** options and select a destination folder for the asset. To copy the asset to the destination folder, click **Copy**. Click **Move** to move the asset to the new destination.

**Note:** If you copy or move an asset to the **Shared** folder, you are prompted to set user group permissions to the asset or folder.

## Copy and paste images, pictures, web pages, and text

You can now copy content (such as an image, media, web page, or text) and paste it directly into an IBM Planning Analytics Workspace book. The software automatically identifies the type of content you pasted and converts it to the appropriate widget, simplifying the process to add content.

To copy paste content directly into a book, copy the content and open a book in **Edit** mode. Click anywhere in the book and use **CTRL+V** (Windows) or **CMD+V** (Apple Mac) to paste the copied content.

You can copy paste:

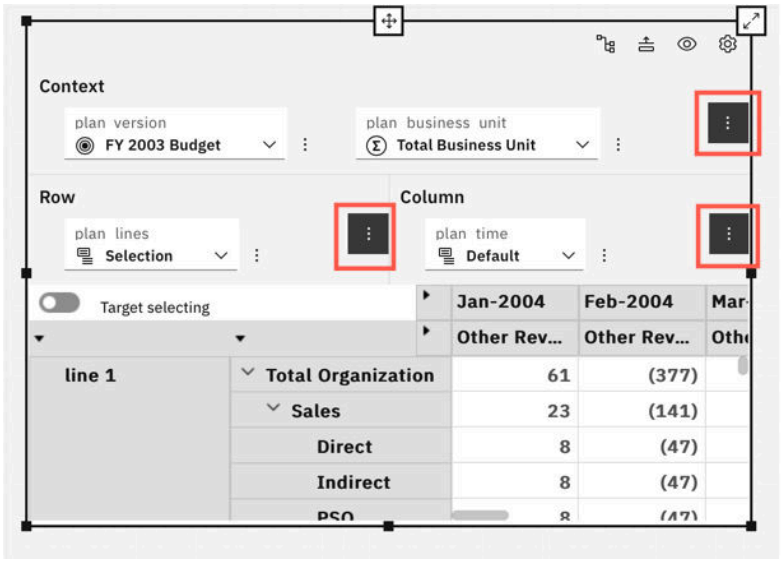
- Images from your computer, online images from your browser, or links to image files
- URLs to media format (such as online videos)
- URLs to web pages
- Text

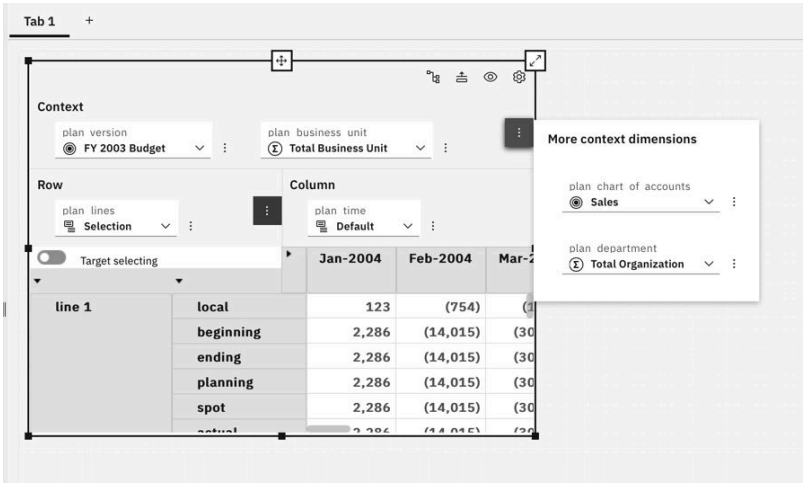
You can also continue to add content to your book by using the **Widgets** icon in the side bar. For more information, see Add pictures, media, and web pages.

## View all and View less links replaced by a menu in new experience view

The **View all** and **View less** links, which were used to see and hide dimensions from the view, have been replaced by a new menu.

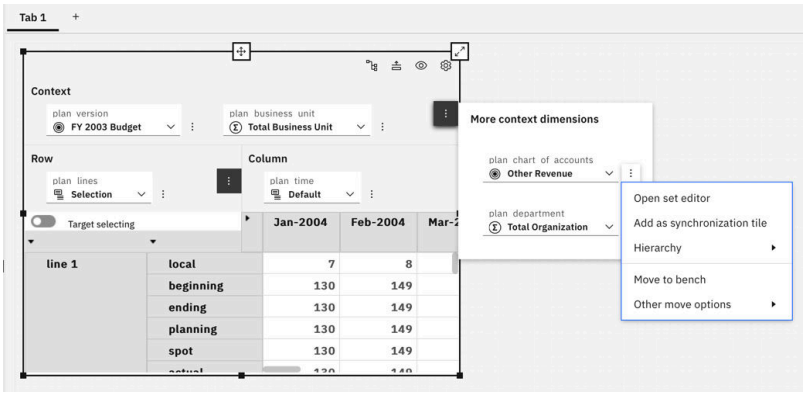
Depending on the number of dimensions in a view or the view's size, you might not be able to see all the dimensions supported by the view. Click the menu icon in the **Context**, **Row**, or **Column** areas, to see dimensions that are hidden from display.




You can move dimensions from this menu to the **Context**, **Row**, or **Column** areas and vice versa. You can also move dimensions within the menu.

Click the dimension's menu to see options such as **Open set editor**, **Move to bench**, **Other move options** and more.



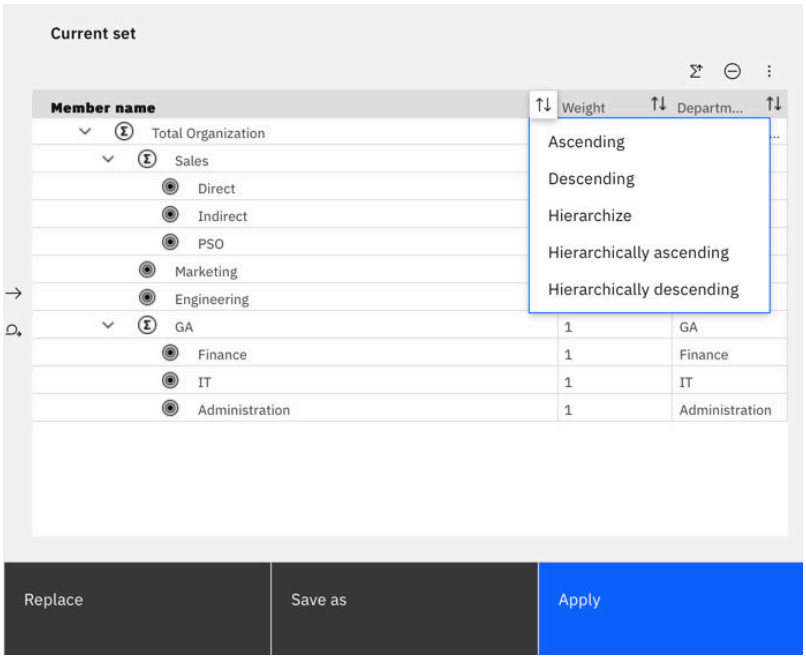

For more information, see New experience view and set editor.

## Sort members, attributes, and properties in the new set editor

The new experience set editor now supports sorting of members, attributes, and properties.

**Note:** To use the new experience view and set editor, you need to enable the feature under **Features** in **Administration**. For more information, see Manage features.

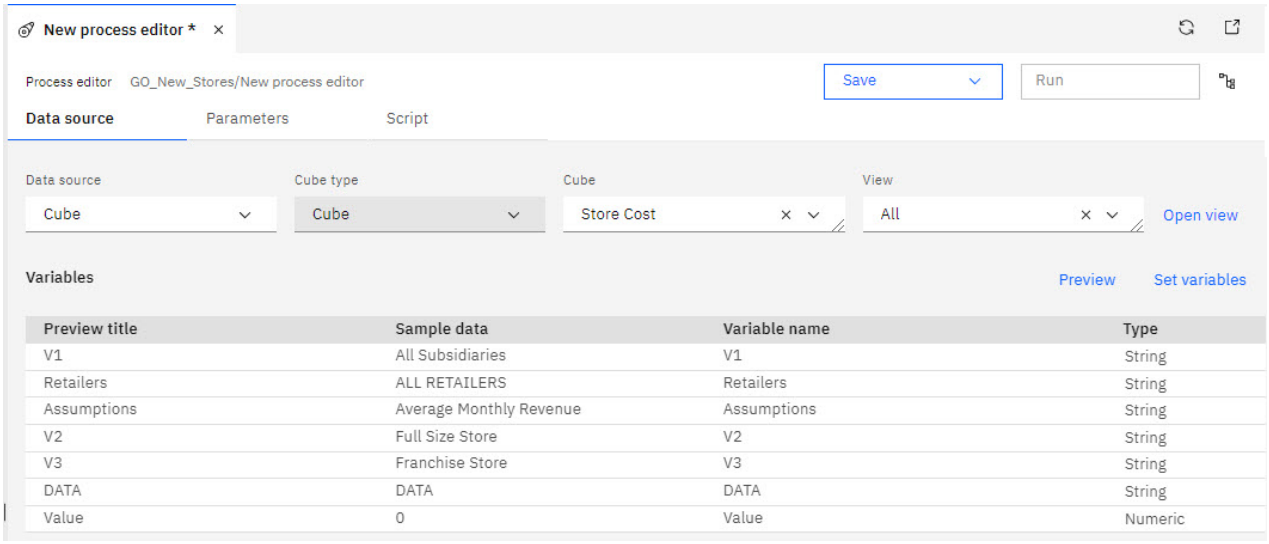
Click **Sort** next to the member, attribute, or property name under **Current set** and select how you want to sort the column (in ascending, descending, hierarchically ascending, or hierarchically descending order).




Click **Hierarchize** to revert any sorting you applied to the member's column.

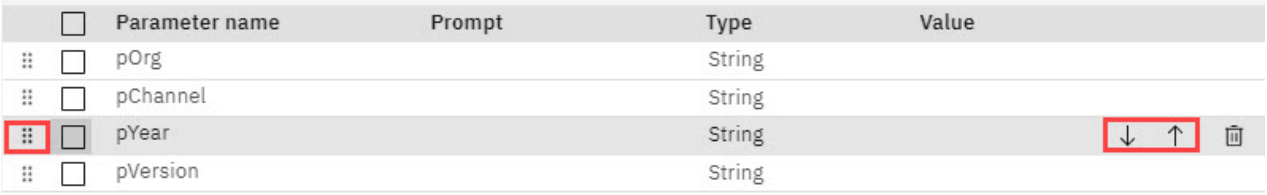
## New process editor

As part of the ongoing effort to standardize the Planning Analytics Workspace user experience, the process editor has been updated. Note that any existing process editors that were previously saved in a modeling book or workbench remain available using the previous interface in the modeling book or workbench.

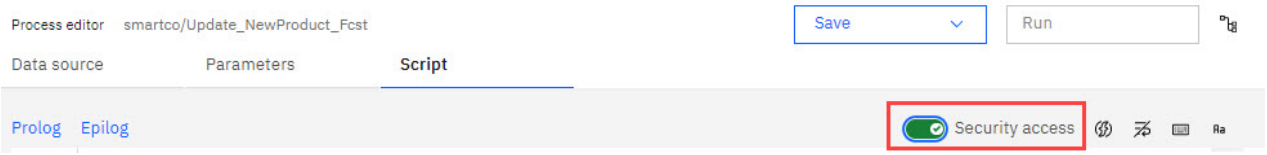



The new editor also includes several usability improvements.

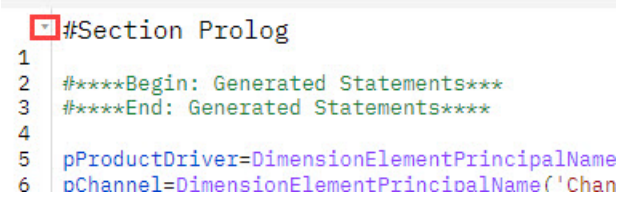
You can change the order of parameters on the **Parameters** tab. Click a parameter handle and drag to a new location or select a parameter and use the **Up** and **Down** arrows to move the parameter.




You can enable or disable the permission for a process to modify security data with the **Security** access switch directly on the **Scripts** tab.




You can click the **Expand/Collapse** arrow next to any section on the **Scripts** tab to expand or collapse a section.



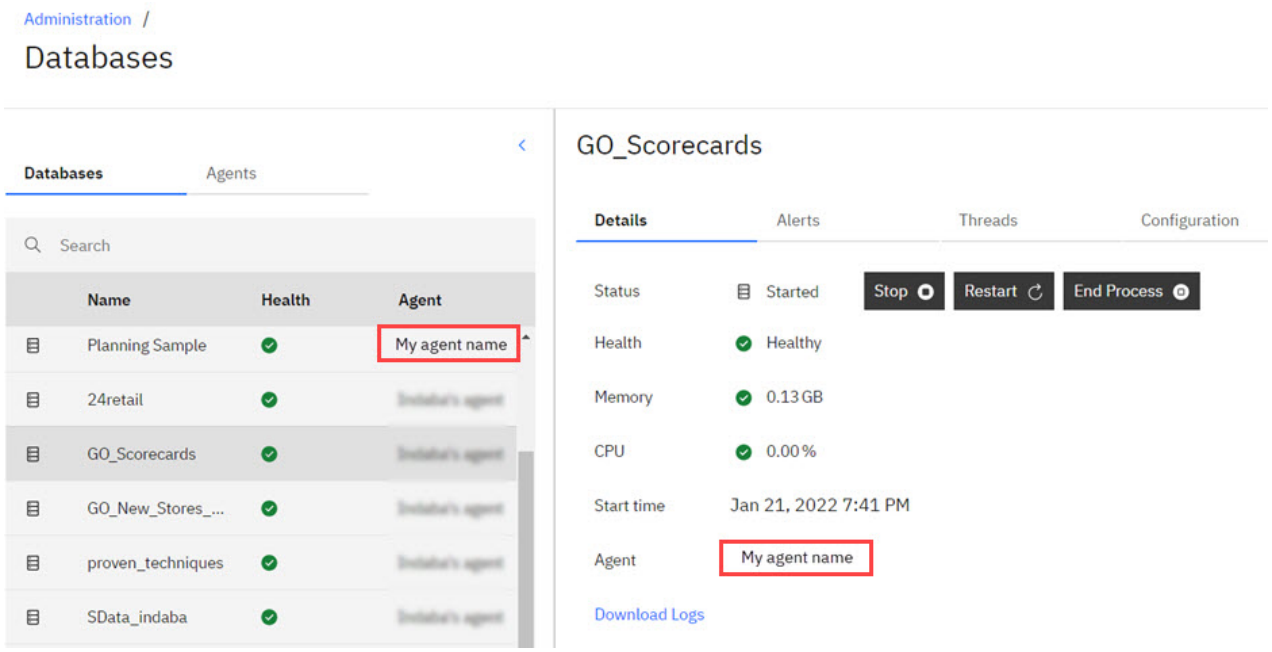

You can click the **Find in tree button** to immediately locate the process in the Databases tree.




## Show agent label instead of IP address in Planning Analytics Administration (Local only)

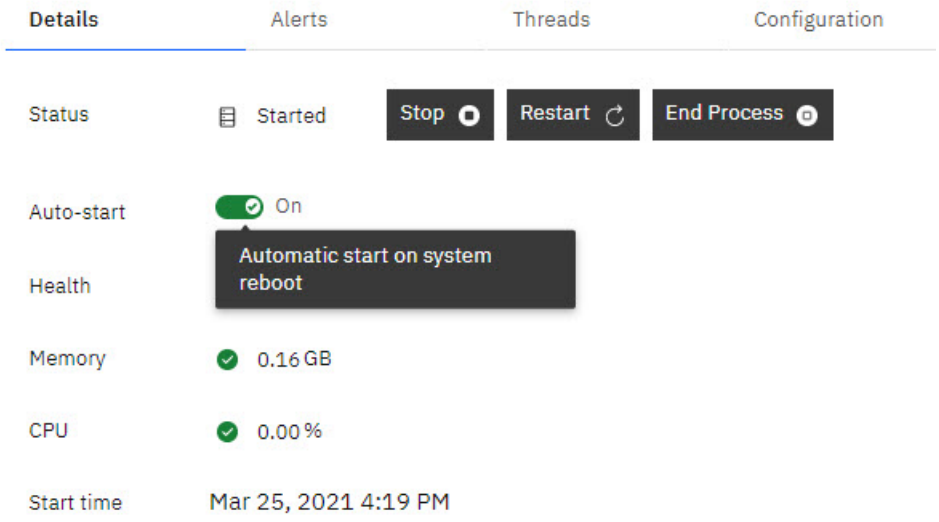
A new property to the `bootstrap.properties` file lets you specify an agent label, which is displayed in place of the agent IP address in Planning Analytics Administration.

To use this new capability, add the property AGENT_LABEL=*"agent_name"* to the `bootstrap.properties` file for your agent. When this property is set, you'll see the agent name rather than the IP address in Planning Analytics Administration.




## Auto-start option for databases (Cloud only)

The **Details** tab on the database administration page now includes an **Auto-start** option to automatically start a database when the data tier is restarted during regular monthly maintenance.




It's possible that you don't want some of your databases to start when the data tier is restarted, while other databases are essential and must be started along with the data tier. Use the **Auto-start** option to control the behavior of individual databases. By automatically starting only essential databases upon data tier restart, you can conserve system resources and reduce database startup time.

You can always manually start a database from the Databases administration page.

**Note:** If a database has a space in its name and the **Auto-start** option is **disabled**, there is a known issue that prevents the database from being manually started after the data tier is restarted. To work around this issue, enable the **Auto-start** option then start the database. This issue will be fixed in an upcoming release.

# 2.0.72 - What's new, December 20, 2021

IBM Planning Analytics Workspace 2.0.72 contains no new features. This version is provided solely to address vulnerabilities in Apache Log4j, which is used by Planning Analytics Workspace as part of its logging infrastructure.

For full details on Log4j vulnerability, see Security Bulletin: IBM Planning Analytics 2.0: Apache log4j Vulnerability (CVE-2021-44228).

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

# 2.0.71 - What's new, December 14, 2021

Learn about new features and known issues in version 2.0.71 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Recommended cohorts

For reference purposes, here are the most recent and recommended cohorts for Planning Analytics Workspace 2.0.71:

- **Planning Analytics for Microsoft Excel 2.0.69**. For more information, see Download Planning Analytics v2.0 - IBM Planning Analytics for MS Excel Release 69 from Fix Central.
- **IBM Planning Analytics Local 2.0.9.10**. For more information, see IBM Planning Analytics Local 2.0.9.10 is now available on Fix Central.

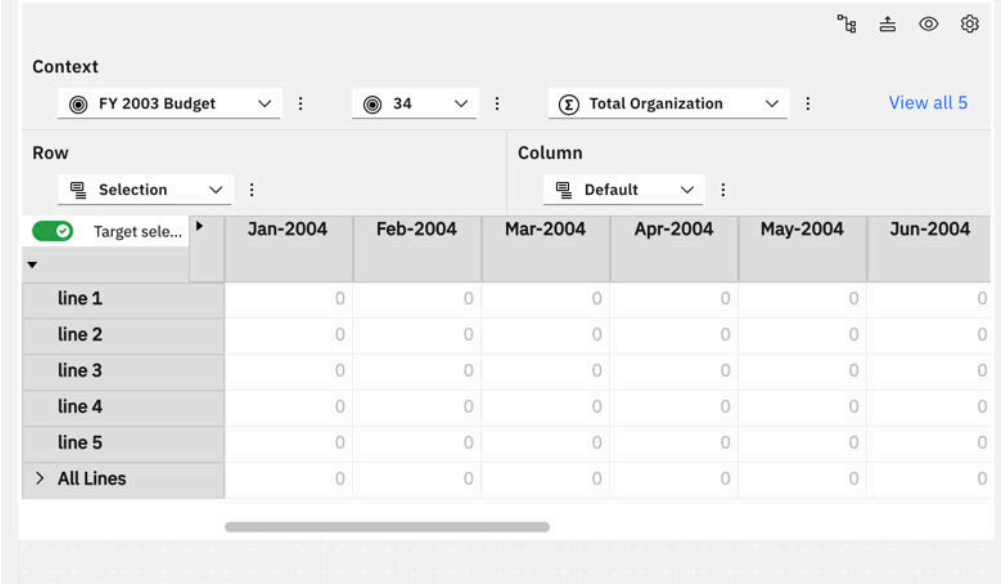
## Timeline of previous Planning Analytics releases

For a comprehensive listing of Planning Analytics component releases by month and year, see Timeline for IBM Planning Analytics Releases. You can view versions and release months for the following components:

- Planning Analytics Local TM1 Server
- Planning Analytics Workspace
- Planning Analytics for Microsoft Excel
- Planning Analytics Spreadsheet Services/TM1 Web

# New experience view and set editor

A new experience view (or exploration) and set editor are now available in **Books**. With several improvements and enhancements, the new view and set editor offer flexible options to view, analyze, and organize data.



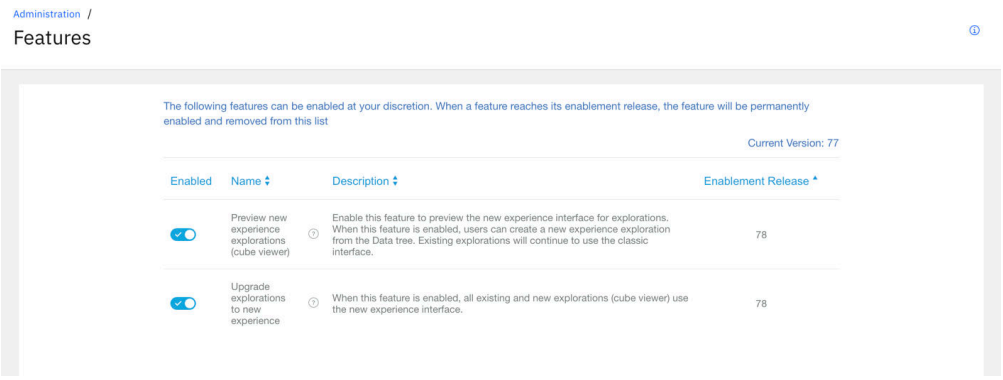

A video walk-through of the new experience view and set editor is available on the IBM Business Analytics Community.

Use the following links to navigate to the relevant section in this topic:

- Enabling the new experience view (preview and upgrade)
- Using the new experience view
- Using the new set editor

## Enabling the new experience view

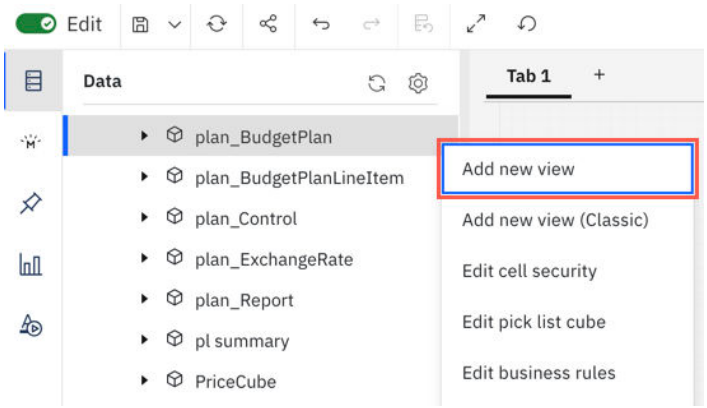
You can preview or upgrade to the new experience view under **Features** in **Administration**. For more information, see Manage features.




Previewing the new experience view:

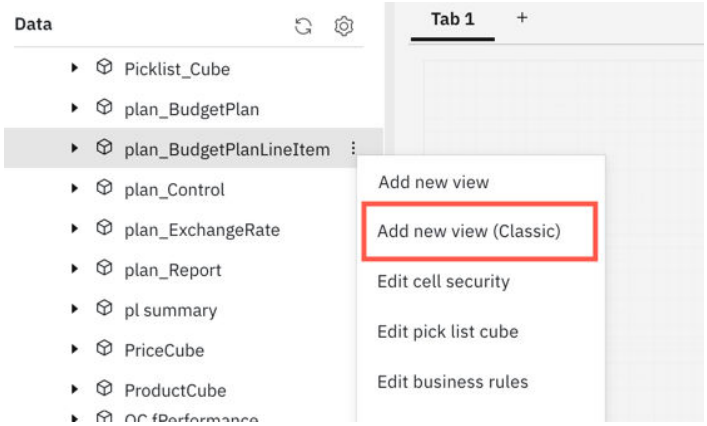
In preview, you can explore and compare the features of both the new experience and classic view interfaces.

After the feature is enabled, open a book in **Edit** mode and right-click the cube in the data tree that contains the data you want to use. Select **Add new view** to use the new view and set editor.




**Note:** You can access the new set editor only through the new view.

To use the classic view, click **Add new view (Classic).**




Upgrading to the new experience view:

When you upgrade to the new experience view, all existing and new views in **Books** use the new experience interface. Any existing views that use the classic interface are converted to the new experience but retain the look (colors and font) of the classic interface.

**Note:** Views and set editors in the modeling workbench and IBM® Planning Analytics for Microsoft™ Excel are not changed.

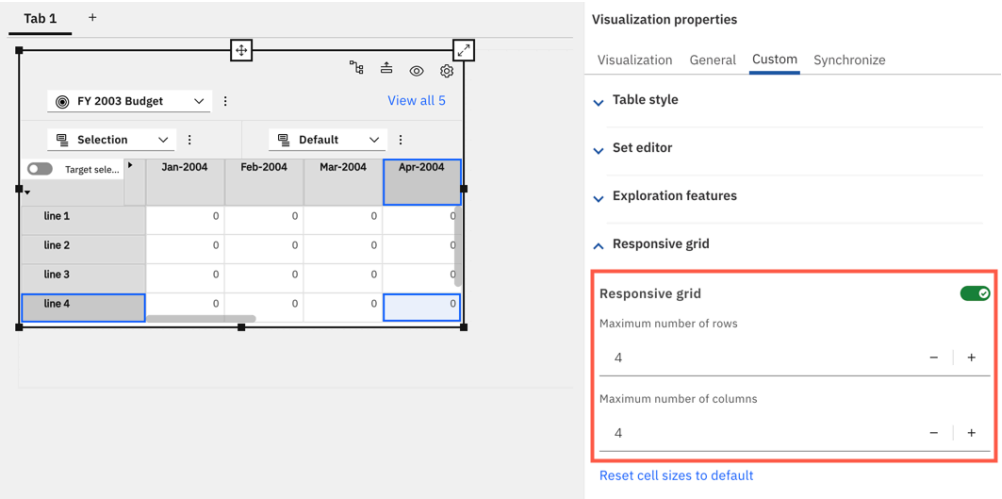
Upgrading views to the new interface also upgrades set editors but only those that are opened from the new experience view. If you add a set editor as a widget to a book from the data tree, the set editor continues to use the classic interface.

## Using the new experience view

The new view has several improvements and enhancements that make features more accessible.

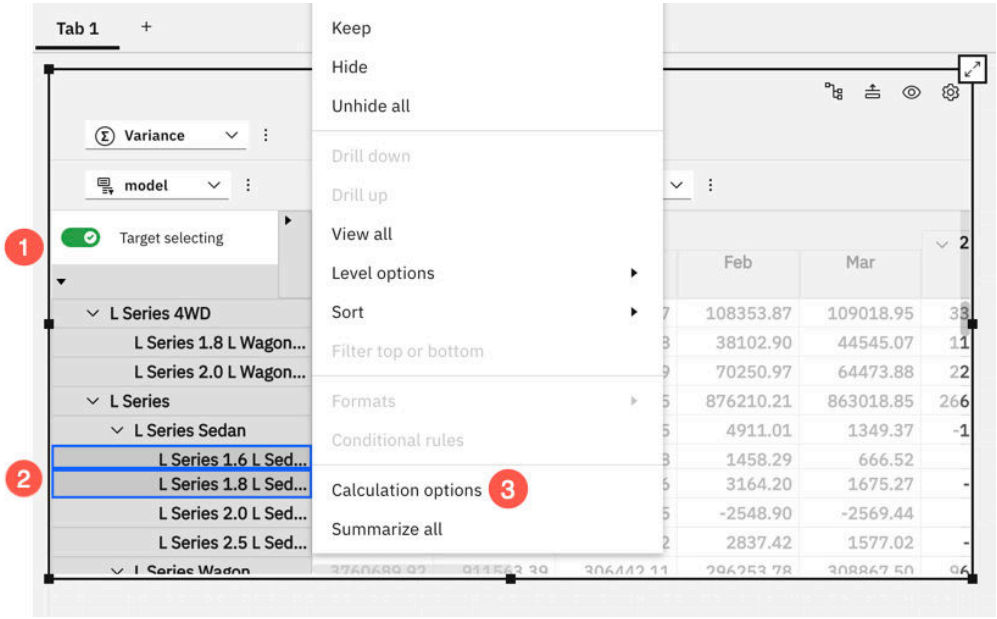
- **Responsive grid property**

  A new **Responsive grid property** property makes the view responsive to size changes. Enable the property to set a fixed number of columns and rows. The view displays the fixed number of columns and rows when you resize the grid.



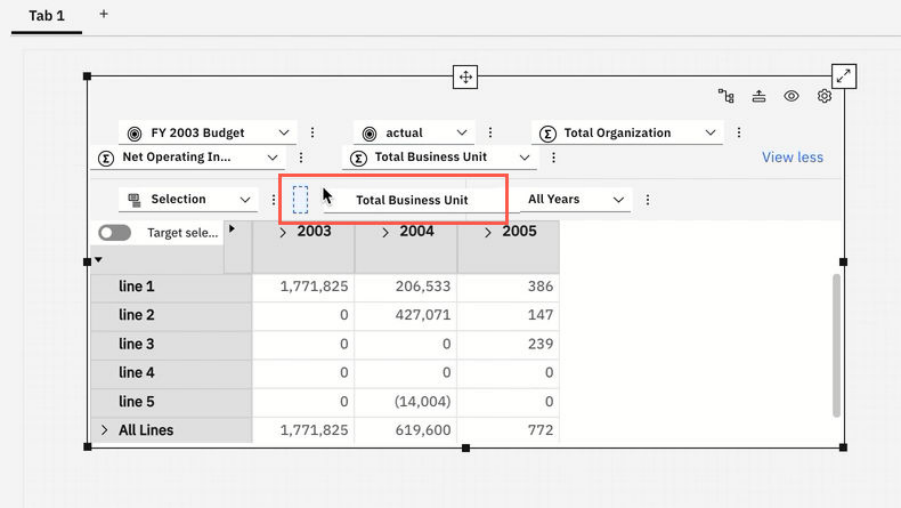

- **Target selecting toggle for asymmetric calculation**

  The new **Target selecting** toggle is used to create an asymmetric calculation. You no longer need to select the individual button on the row or column headers to create an asymmetric calculation. Instead, enable **Target selecting** and select the members that you want to use in the asymmetric calculation. Right-click a selected member and click **Calculation options**.




- **A more guided and intuitive experience**

  The new view's improved design offers a more guided experience without compromising its capabilities. For example, the new view now guides you to the drop zone when you drag a dimension to a different location in the grid.

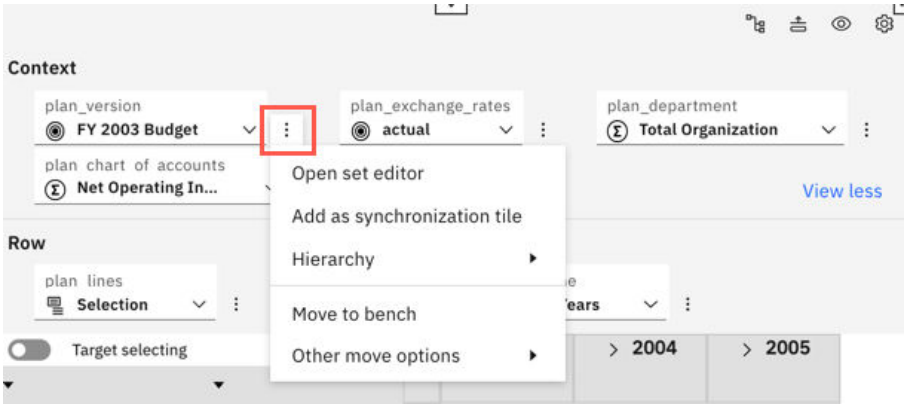



Use the icons in the grid  to:

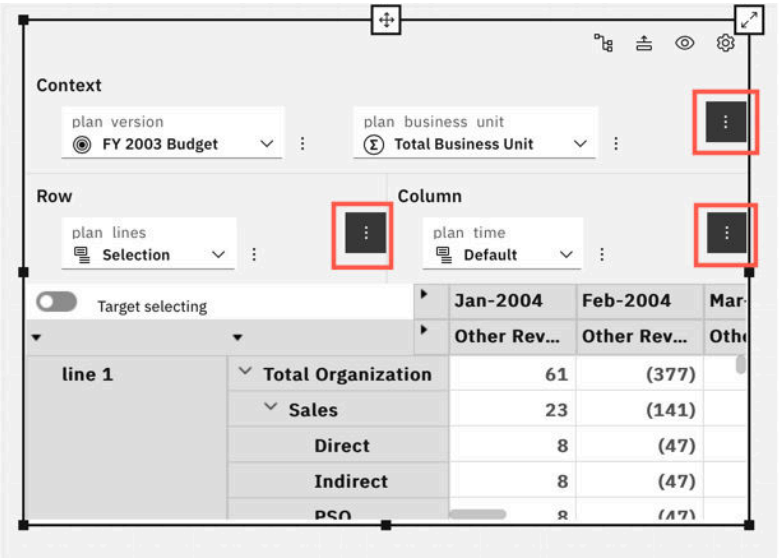
- See the cube in the data tree.
- Expand or collapse the overview.
- See benched dimensions.
- Select hierarchy, iconography, context information, and section headers to customize your view.

Each dimension tile in the grid now has a menu icon next to it from which you can:

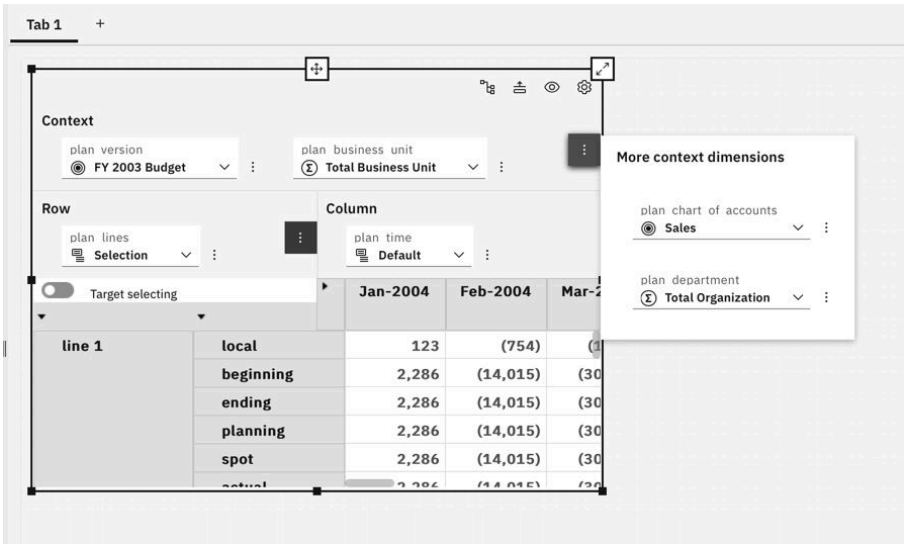
- Open the set editor.
- Add the dimension as a synchronization tile.
- Remove, replace, or add related hierarchy.
- Move the dimension to the bench.
- Manually move the dimension right or left, or to a row, column, or context.



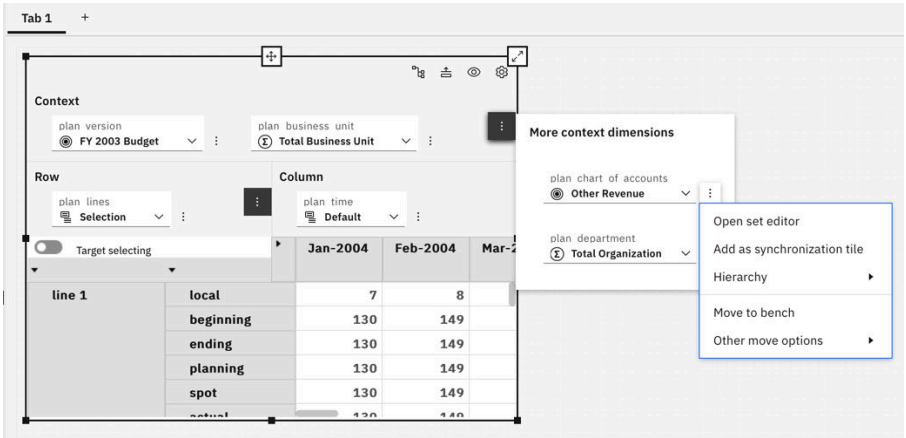

Depending on the number of dimensions or the size of the view, all dimensions might not display. To see dimensions that are hidden from the view, click the **More dimensions** menu in the **Context**, **Row**, or **Column** area.




From the **More dimensions** menu, you can drag dimensions to the **Context**, **Row**, or **Column** area and vice versa. You can also move dimensions that appear within the **More dimensions** menu.




You can click the menu next to the member's name to open the set editor, move the dimension to the bench, see move options, and more.




- **Setting properties in the new view**

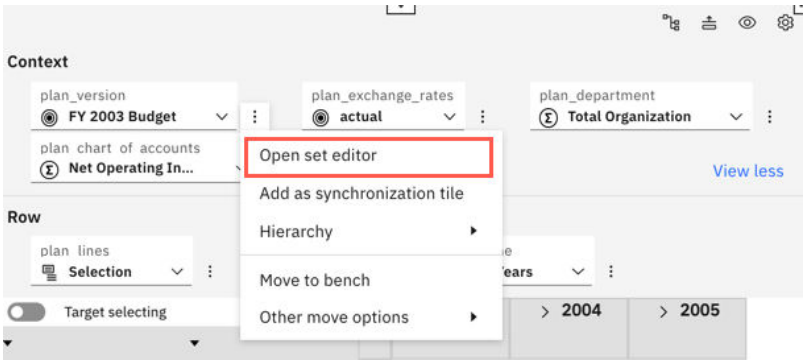
Like in the classic view, you can set properties for the new view by clicking **Properties**.

To offer more formatting flexibility in the new view, the **Themes** property is no longer available.

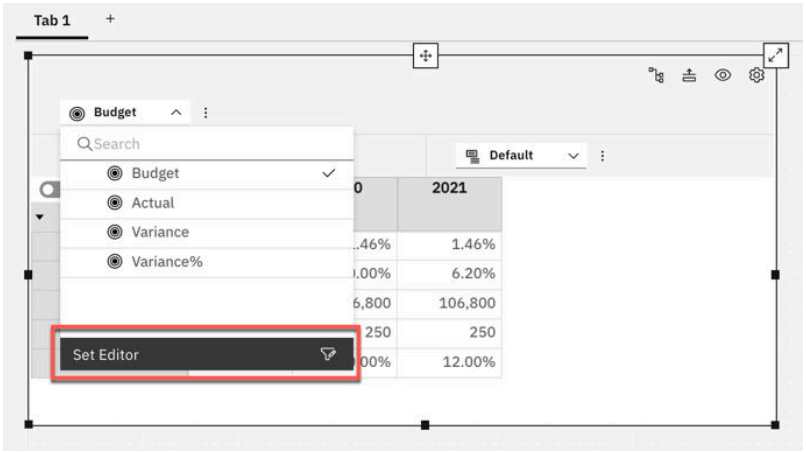
**Note:** If you save a **Book** with a new view and reopen it, the **Book** displays the new view. However, if you disable the new view feature and reopen the **Book**, you see the classic view.

## Using the new set editor

The new set editor can be accessed only through the new view. To open the set editor, click the menu icon next to a dimension in the view and select **Open set editor**.



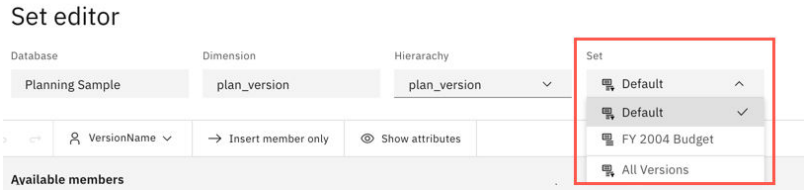

You can also double click the dimension tiles or click the **Set Editor** icon in the dimension menu to open the set editor.




As with the new view, the new set editor offers an improved flow, a more intuitive experience, and flexible options to edit your set.

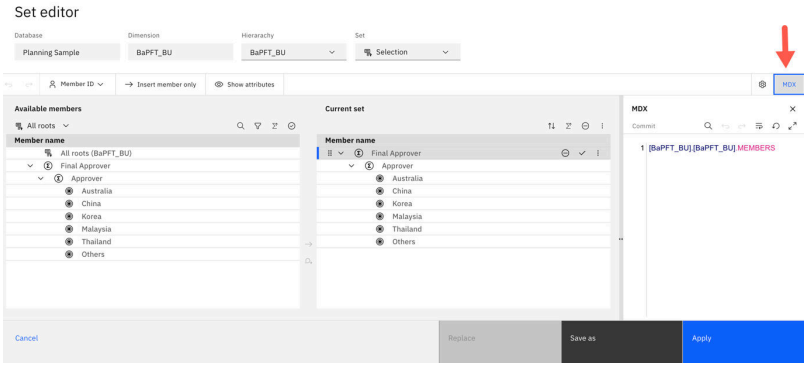
- **Set menu**

  With the **Set** menu, you can now change the set from within the set editor.




- **Improved MDX panel**

  Click **MDX** to see the **MDX** panel and script. You can edit the script in the panel and click **Commit** to apply your changes to the current set.

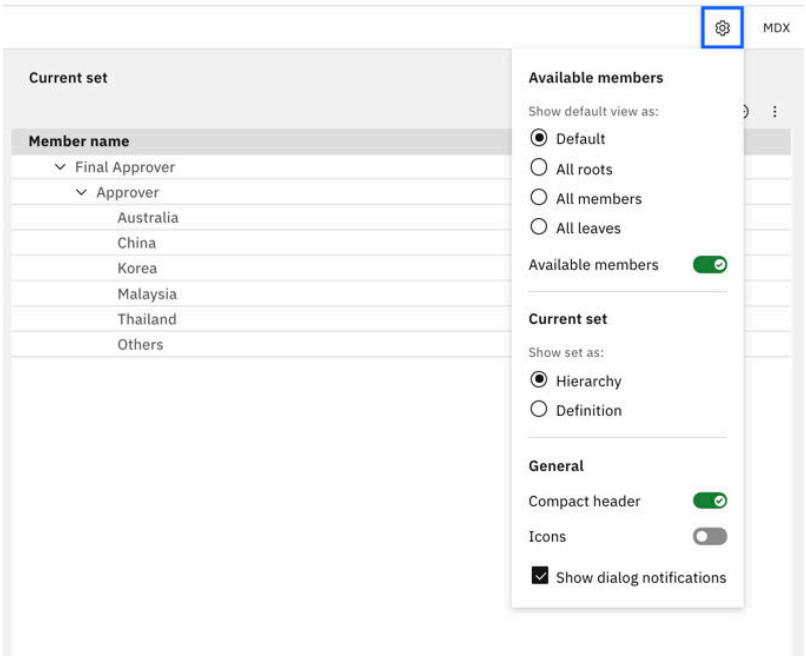



- **Settings menu**

  The **Settings** menu offers options to customize the display for your set.

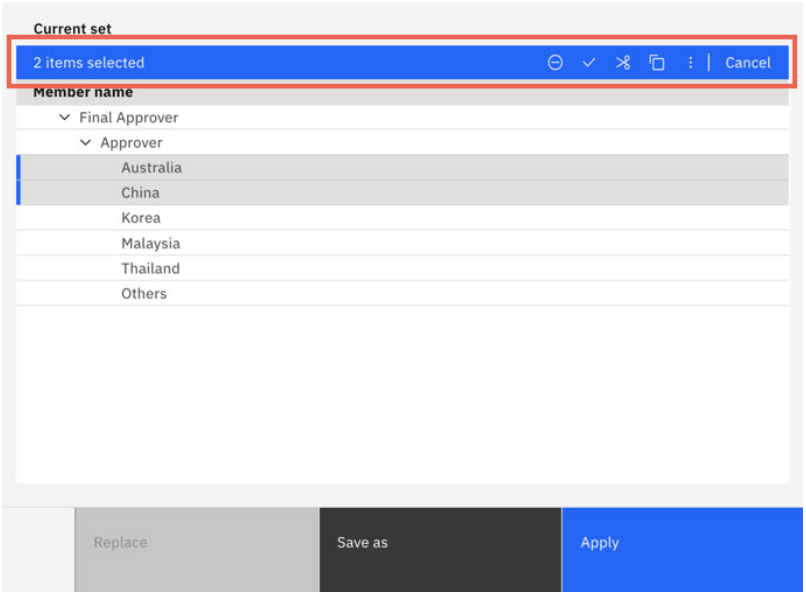
  Enable **Compact header** under the **General** section to reduce the size of the set editor header and see more members.

  You can also remove icons from all members for a simplified display with the **Icons** property under **General**.




- **Multi-select activity display**

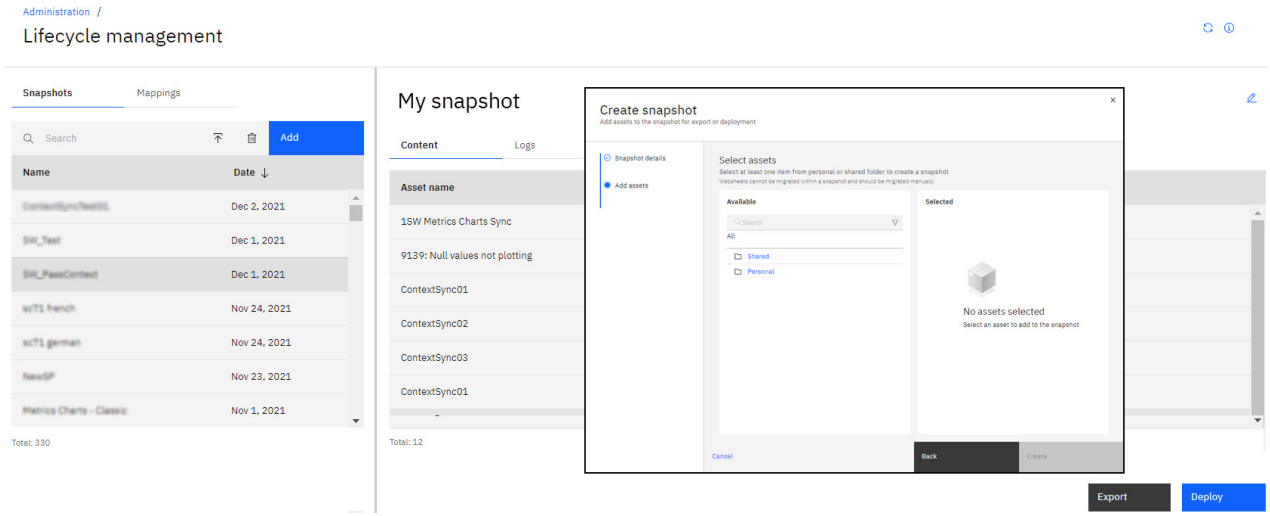
  When you select multiple members in **Available members** or **Current set**, an activity bar now displays with your selection information and available options.




## New UI and functional improvements for Lifecycle Management

A simplified user interface makes it easier to create and manage snapshots.

The new interface also provides a look and feel that is consistent with other Planning Analytics Workspace components.




Here are some of the changes you'll notice in the new Lifecycle Management:

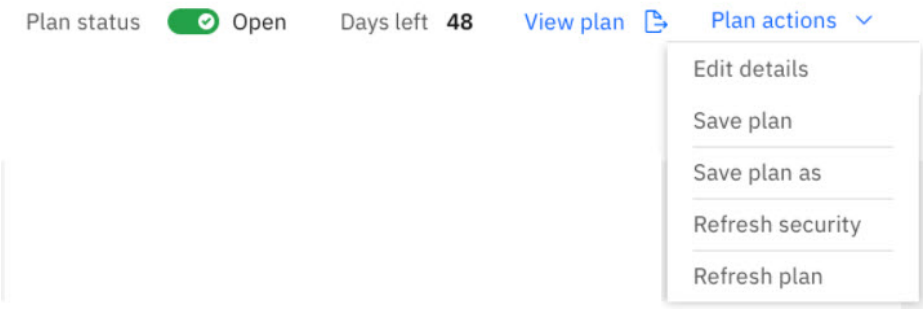
- You no longer have to manage a cart for your assets. Selecting an asset automatically adds the asset to your snapshot.
- You can now select a folder and all its contents to include in a snapshot. This is an improvement over the previous Lifecycle Management, where every asset in a snapshot had to be selected individually.
- You can now include content from both the Shared and Personal folder in a single snapshot.
- Moving a snapshot to a target environment is now called **Deploy** rather than **Migrate** in the user interface.

## New header for plans and applications administration page

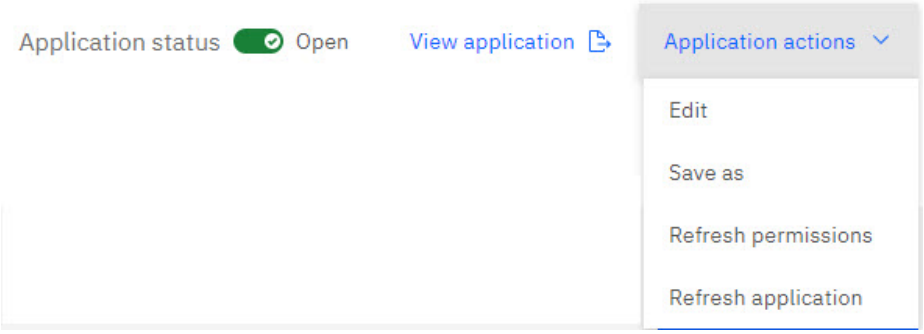
The header section of the administration page for plans and application now includes the ability to quickly open a different plan or application. The header also provides direct access to common administration tasks.

Click ^ next to the plan or application name to open a different plan or application.

Use the options on the plan header to open or close the application, view the plan in contribution mode, or apply any of the plan actions. The **Days left** indicates the number of days left until the last due day for any step in the plan.



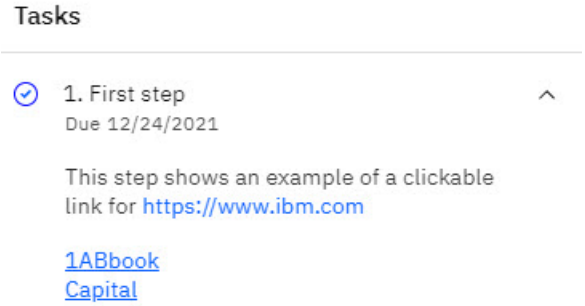

Use the options on the application header to open or close the application, view the application in contribution mode, or apply any of the application actions.




## URLS in plan steps or application sections are now active links

If the description for a step in a plan or a section in an application contains a properly formed URL, the URL is now presented as a clickable link.

Clicking the link opens the destination in a new browser tab.

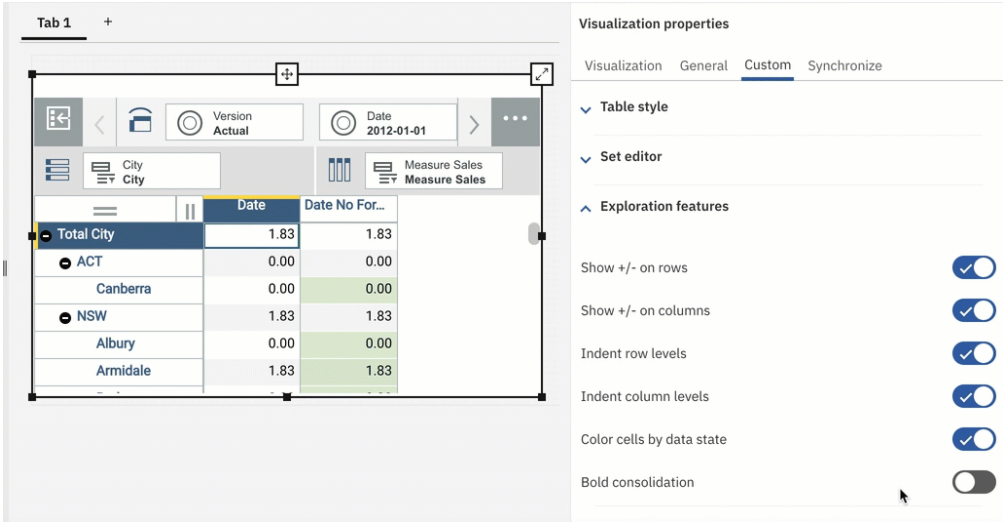


## Enable or disable consolidation formatting

You can now enable or disable the bold formatting for consolidated values in an Exploration.

Consolidation formatting can be enabled or disabled only in **Edit** mode.

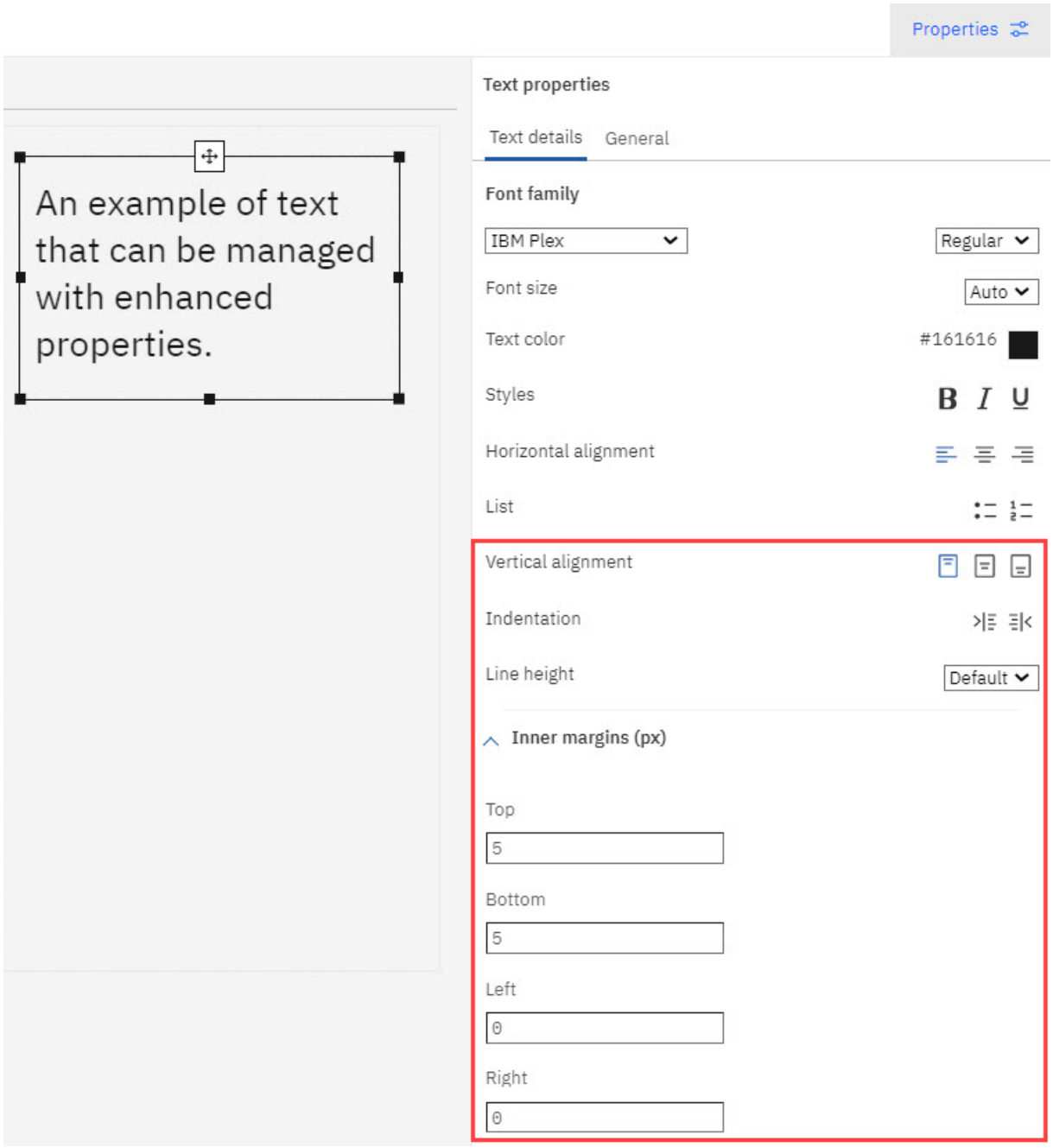
Under **Properties**, go to **Custom** and expand **Exploration features**. To see consolidated values in bold, enable the **Bold consolidation** property. If you disable the property, all values in the Exploration are displayed in regular text.




## Enhanced text formatting

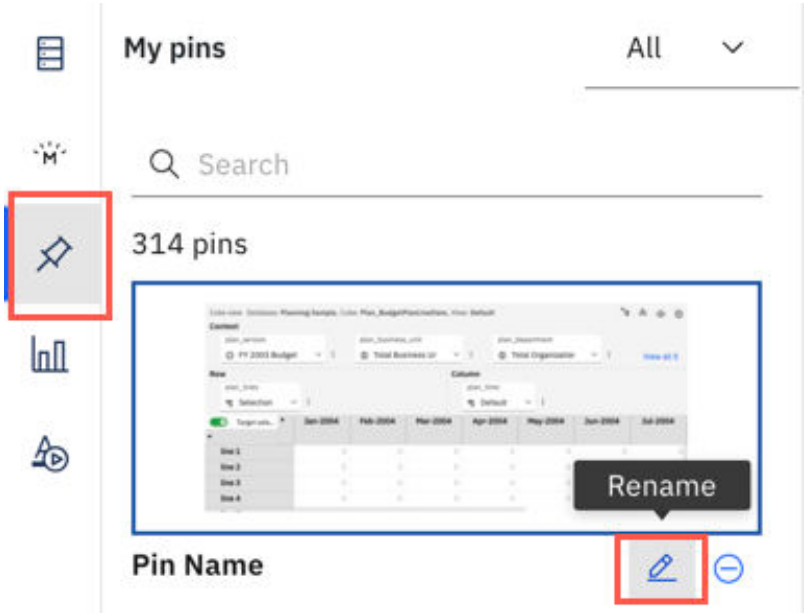
You can now apply properties that manage the vertical alignment, indentations, line height, and inner margins for text within a text widget.

To use the new properties, click the text inside a text widget while in **Edit** mode, then click **Properties**. Click the **Text details** tab to see and use the new text properties.


Properties
An example of text that can be managed with enhanced properties.
Text properties
Text details
General
Font family
IBM Plex
Regular
Font size
Auto
Text color
#161616
Styles
Horizontal alignment
List
Vertical alignment
Indentation
Line height
Default
Inner margins (px)
Top
5
Bottom
5
Left
0
Right
0

### Rename pins

You can now rename a pin when you are in **Edit** mode by clicking the **Rename** icon next to the name of the pinned item.




For more information, see Pin items.

## 2.0.70 - What's new, November 16, 2021

Learn about new features and known issues in version 2.0.70 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

### Recommended cohorts

For reference purposes, here are the most recent and recommended cohorts for Planning Analytics Workspace 2.0.70:

- **Planning Analytics for Microsoft Excel 2.0.69**. For more information, see Download Planning Analytics v2.0 - IBM Planning Analytics for MS Excel Release 69 from Fix Central.
- **IBM Planning Analytics Local 2.0.9.10**. For more information, see IBM Planning Analytics Local 2.0.9.10 is now available on Fix Central.

### Timeline of previous Planning Analytics releases

For a comprehensive listing of Planning Analytics component releases by month and year, see Timeline for IBM Planning Analytics Releases. You can view versions and release months for the following components:

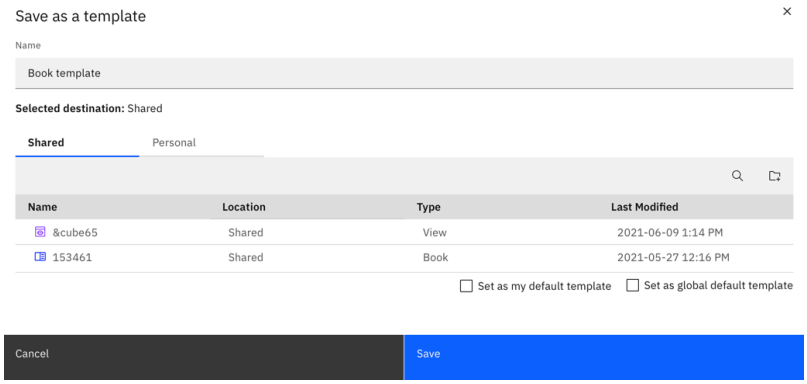
- Planning Analytics Local TM1 Server
- Planning Analytics Workspace
- Planning Analytics for Microsoft Excel
- Planning Analytics Spreadsheet Services/TM1 Web

### Fix list

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Save a book as a template and set it as a default

You can now save a book as a template and set it to be your default. Administrators can also set a book template to be a global default, making it a default template for other users.

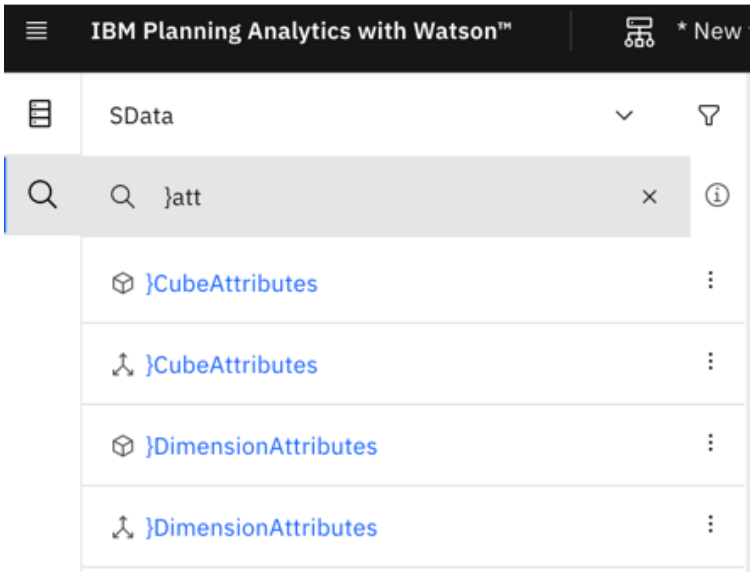



For more information, see Saving a book as a template and setting it as default.

## Modeling workbench search improved to support control objects

In the modeling workbench, you can now search for control objects.

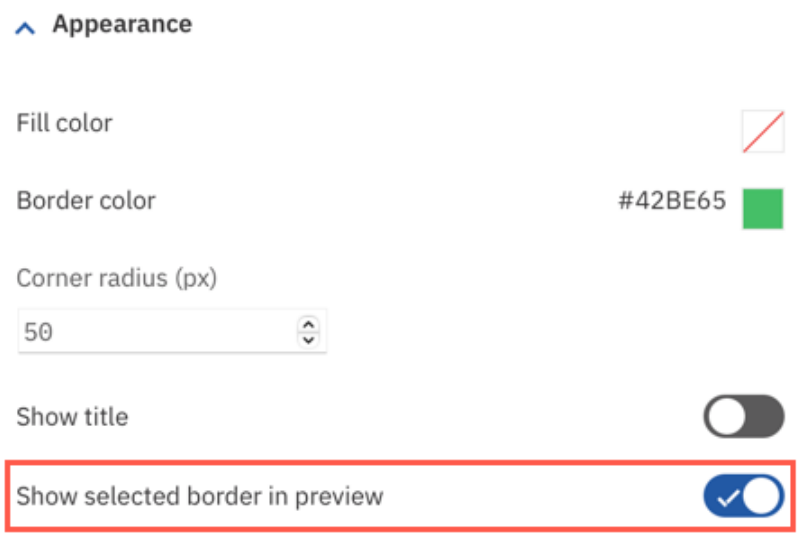
In the **Search** field, prefix the search term with } to search for a control object. For example, `}Capabilities`. The search looks for items that contain the string that follows }.




For more information, see Using database-level search in a modeling workbench.

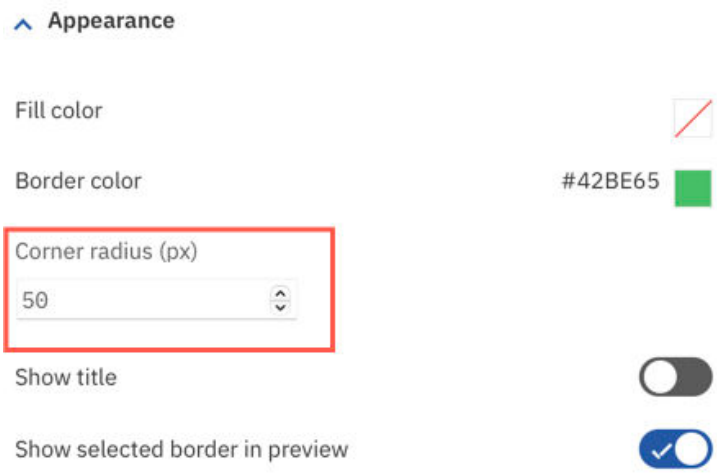
## New property added for item borders

A new property, **Show selected border in preview**, is now available for items (widgets, visualizations, and explorations). With this property, you can enable or disable borders around items when they are in consumption mode.




For dynamic items like visualizations and explorations, the property is enabled by default. For all other items (widgets such as Text, Image, and Shapes), this property is disabled.

You can now also set a corner radius for borders around items.

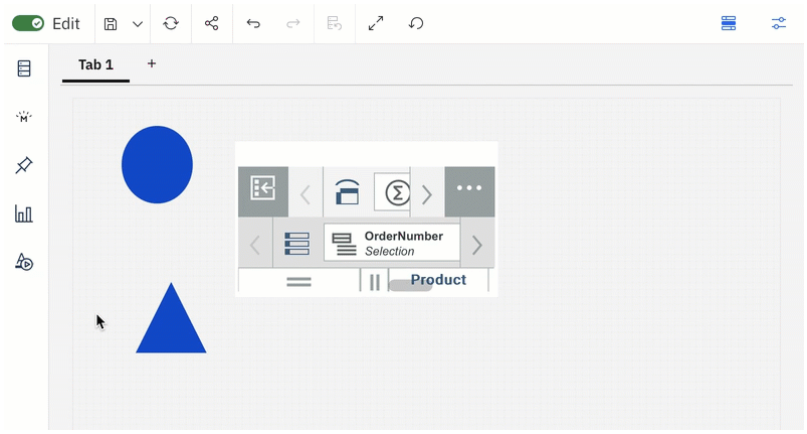



For more information, see Enable or disable borders for items in consumption mode.

## Select multiple items by using mouse drag

You can now select multiple items (widgets, visualization, and explorations) that are on a book tab by using your mouse. Previously, you had to select each item separately.

Click anywhere on the canvas and drag your mouse around the items you want to select. All items within the lasso you created are now selected. With this improvement, you can easily group or move multiple items at the same time.



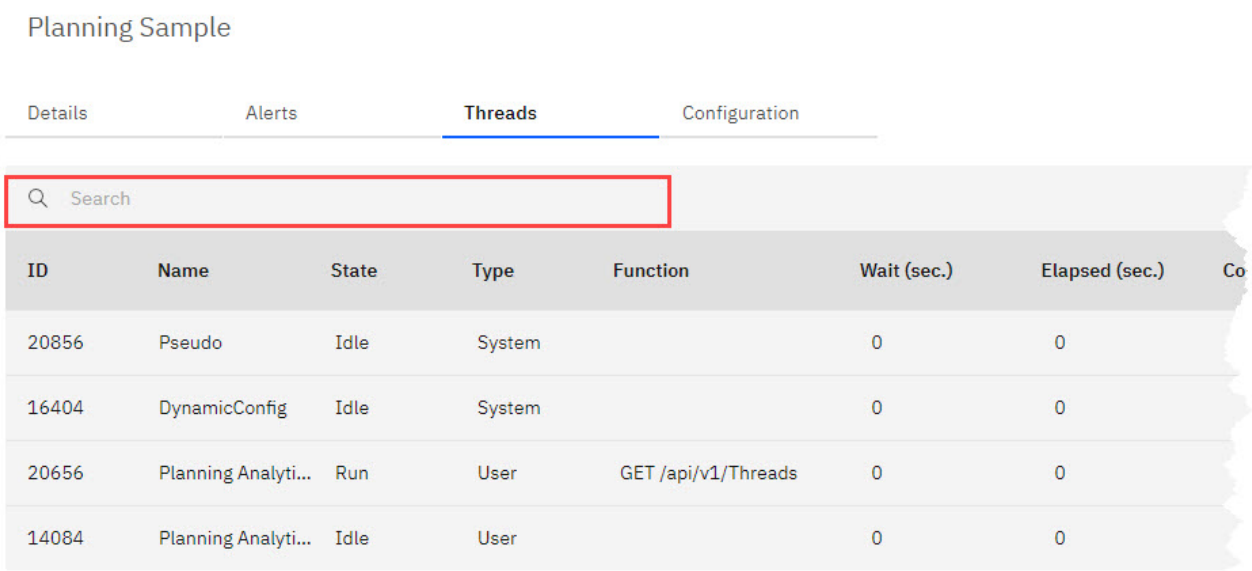

You can select multiple items with a mouse drag in both Edit and consumption mode.

When you are in consumption mode, before you make your selection, enable the Show selected border in preview property for the items you want to select. This makes it easy to see the selected items when you lasso around them.

## Search on multiple words when monitoring thread activity in Planning Analytics Administration

You can now search for multiple words on the database **Threads** tab in Planning Analytics Administration. Separate each word with a comma. For example, **`run,admin,Architect`**.

Search is not case-sensitive.




For more information on monitoring thread activity, see Monitor and manage database thread activity.

## Use custom compact numeric format to display values

You can now append one or more commas to the end of a custom number format pattern to display values in compact format in explorations and visualizations. Each comma present at the end of a format pattern scales the value display by 1,000.

A single comma results in the value being displayed as thousands. Two commas display the value as millions. Three commas display the value as billions.

For example, the format pattern `#,##0.0,` results in the value 1234567.89 being displayed as 1,234.6. The format pattern `#,##0.0,,` displays the same value as 1.2.

To improve the usability of compact format, you can include literals in the format pattern to display specified characters. You can use these characters to indicate that the value is displayed as thousands, millions, or billions. Literals must be enclosed in single quotation marks.

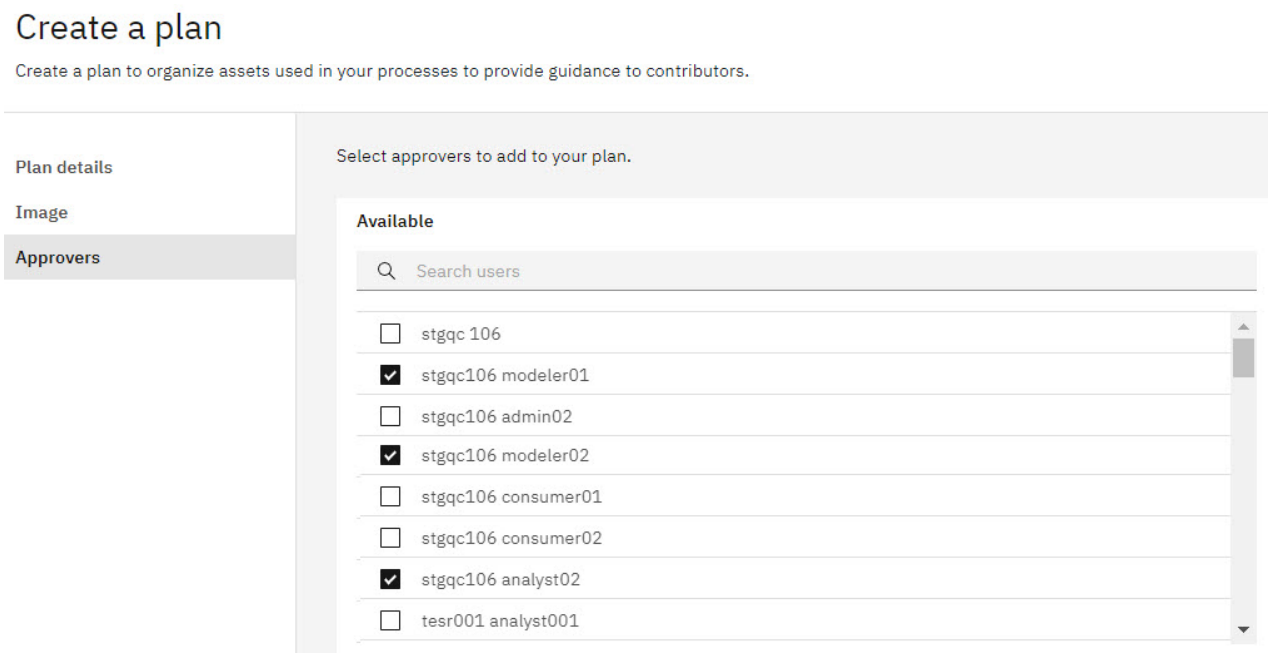
To improve upon the previous example, use the format pattern `#,##0.0,'K'` to display the value 1234567.89 as 1,234.6K. The format pattern `#,##0.0,,'M'` displays the same value as 1.2M.

For more information on setting custom formats, see Setting the data format in a view.

## Use enhanced workflow and capabilities when you create plans and applications

The workflow for creating plans and applications is modified to simplify the creation process.

When you initially create a plan or application, you set the name, description, and image on a page separate from the plan or application template.




You can still modify these details directly on the plan or application at any time.

Additionally, you can now set approvers for a plan or application. Approvers are users who can fully edit a plan or application after its initial creation. An approver has all the capabilities of an administrator for a plan or application, including the ability to submit a plan step on behalf of any group.

For more information on creating plans and applications, see Use plans and applications to organize work.

## Submit a plan step for all groups to which you belong

When you belong to more than one group that is assigned to a plan step, you'll see a new option to **Submit all groups** when submitting a step. Select this option to submit the plan on behalf of all groups to which you belong.

For more information, see Open and contribute to a plan.

## Planning Analytics Workspace Local support for Red Hat Enterprise Linux 8 (Local only)

You can now install Planning Analytics Workspace Local version 2.0.70 or later on Red Hat Enterprise Linux 8.

For details, see Installing Planning Analytics Workspace Local.

## Deprecation notice - modeling books in Planning Analytics Workspace

Planning Analytics Workspace 2.0.70 is the last release that allows modelers to create new Modeling Books.

The ability to create modeling books will be removed in the next Planning Analytics Workspace release. Existing modeling books will continue to be usable. IBM recommends the use of the Modeling Workbench as a replacement for modeling books.

# 2.0.69 - What's new, October 12, 2021

Learn about new features and known issues in version 2.0.69 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

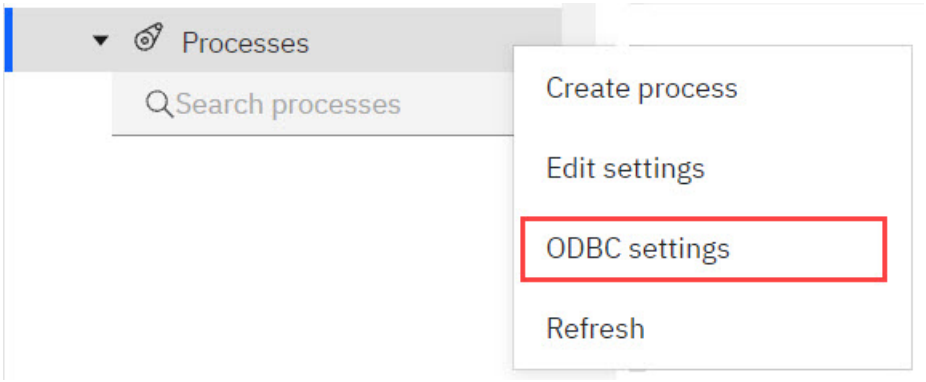
You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Manage ODBC connections for multiple processes on the modeling workbench

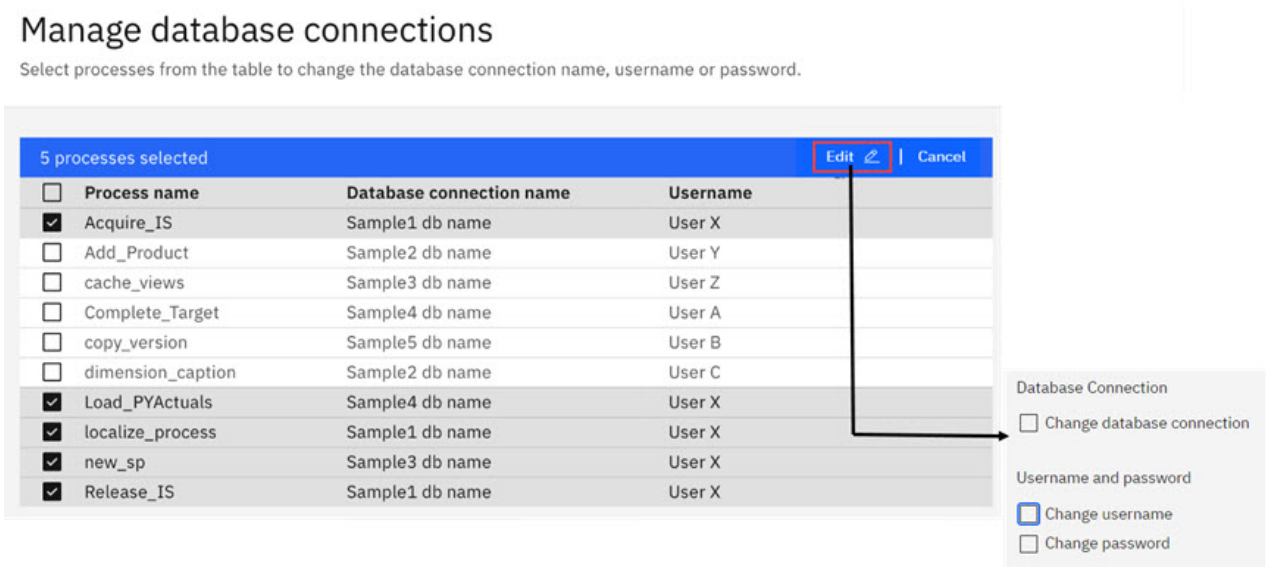
Each process with an ODBC data source includes the declaration of the ODBC connection, username and password as part of the data source connection configuration. When your database contains multiple processes that use the same data source connection configuration, it can be laborious to update each process individually when the connection configuration changes.Use the new connection manager to modify the ODBC connections for multiple processes simultaneously.

**Note:** The connection manager is available only on the modeling workbench. You can't use the connection manager in a modeling book.

To open the connection manager, click the **Actions** icon ⋮ on the Process node in the Databases tree, then click **ODBC settings**.




In the connection manager, you can select which processes you want to modify, then click **Edit** to modify the database connection type, user, or password.

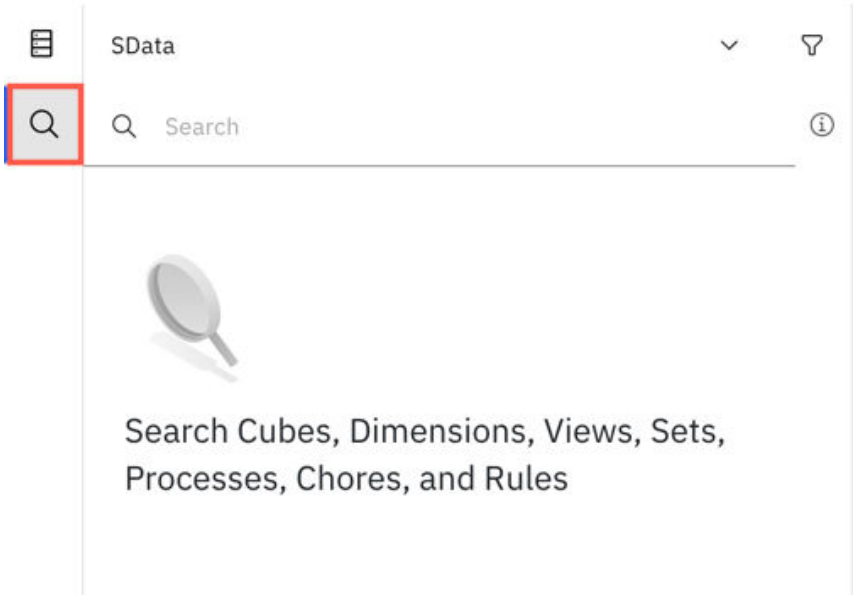



After you save the connection changes, the modified connection configuration is applied and saved to each of the processes you selected.

For more details, see Modify connection configuration for multiple processes that use ODBC data sources.

## New database-level search in the modeling workbench

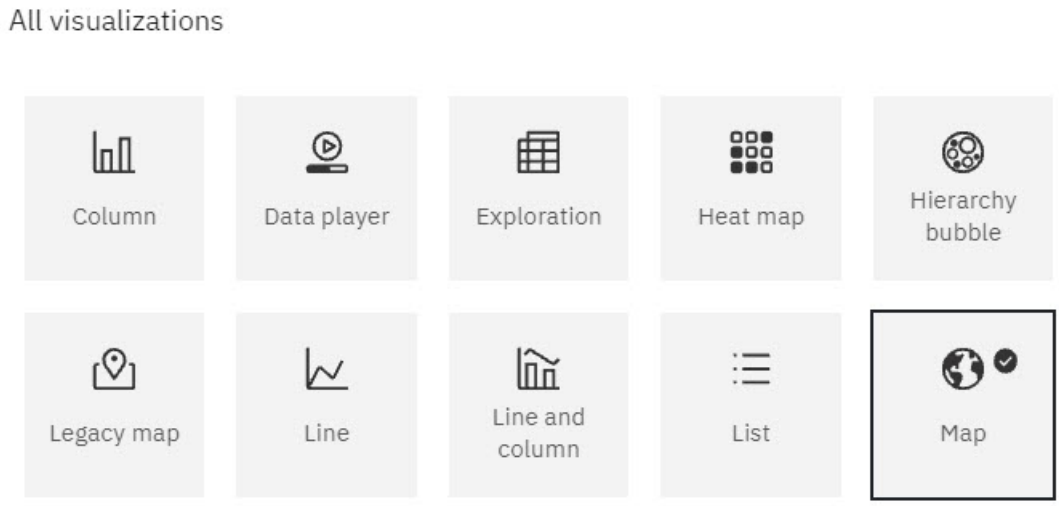
In the modeling workbench, you can now search for data items (cubes, dimensions, views, sets, processes, chores, and rules) in any database. This lets you quickly find a data object and open it in a tab. You can also edit or delete a data item from the within the search panel.




For more information, see Using database-level search in a modeling workbench.

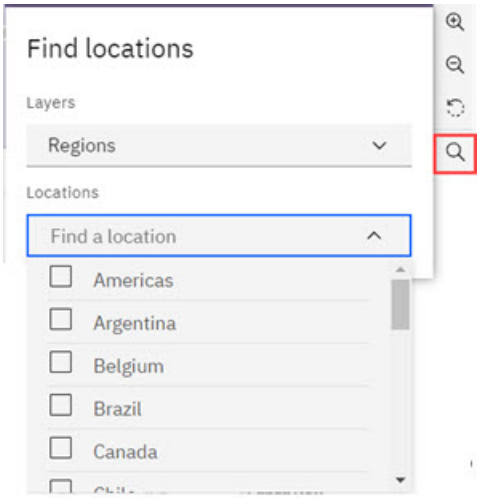
## Use the new Map visualization to view geographic data

A new Map option is now available to create visualizations for data with a geographic component.

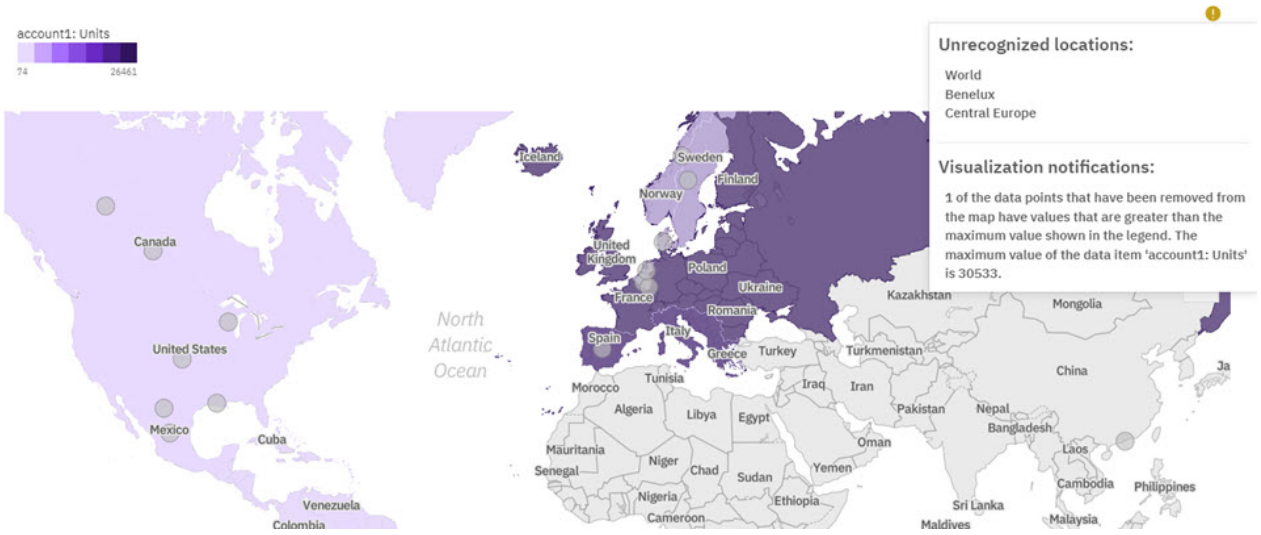



Both the new **Map** and the existing **Legacy map** visualizations are available for use. Any books that contain a Legacy map remain unchanged; Legacy maps are not automatically converted to the new Map visualization.

The new Map visualization provides improvements in geographic dimension identification, general usability, and error warnings. For example, you can easily search for locations directly from the map.




Additionally, if your data contains unrecognizable locations or data outliers, you can view that information while you create or use a Map.




Another advantage of the new Map visualization is that you can use integration with Mapbox to display custom polygons or layers in your maps.

## Reset a book

You can now reset your book to reverse changes you made and go back to the last saved version of the book. Resetting a book is useful if you want to reset all the filters of a book to their initial selections.

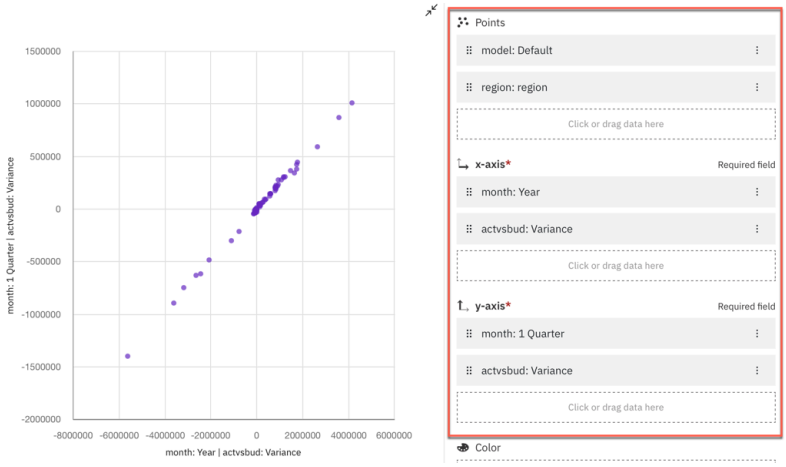
Click the **Reset book** button in the IBM Planning Analytics toolbar to reset book changes.




For more information, see Resetting a book.

## Enhanced Scatter visualization

You can now assign multiple data items to all the fields in **Scatter** visualizations. For example, in this screen capture, there are two data items mapped to each field.




For more information, see Scatter visualization.

## Rollback data entry button updated

Use  to now rollback your data entries. There is no change to the rollback data entry feature and it continues to function as before.

For more information, see Rollback data entry.

# 2.0.68 - What's new, September 14, 2021

Learn about new features and known issues in version 2.0.68 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

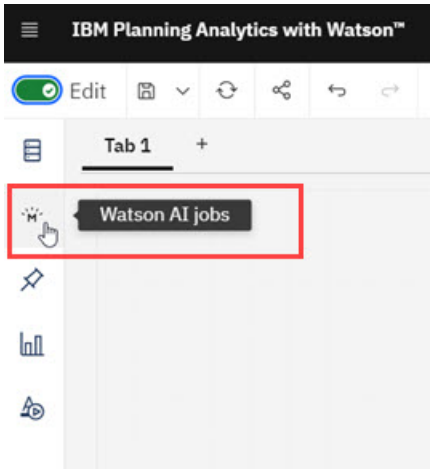
You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Use Decision Optimization with Planning Analytics Workspace (Cloud only)

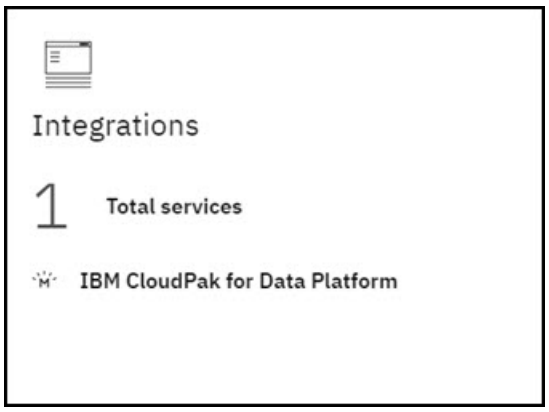
You can now use Decision Optimization (IBM ILOG® CPLEX®) with IBM Planning Analytics Workspace on Cloud to deliver prescriptive analytics capabilities to help you make better decisions and deliver improved business outcomes.

With the Cloud Pak for Data Platform integration configured, and the required services IDs and views in place, analysts can easily view and run optimization jobs within the configured deployment space directly from Planning Analytics Workspace.  Additionally, they can configure action buttons to run any of the jobs of a deployment space, allowing any Planning Analytics Workspace user to run optimization jobs.

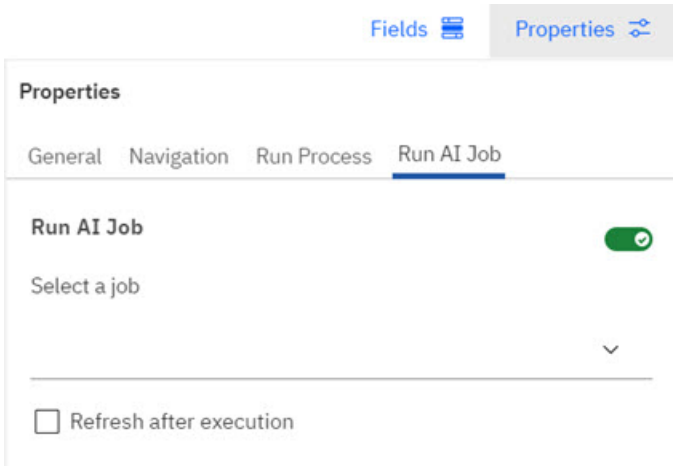
Administrators, analysts, and modelers can access optimization jobs from the new **Watson AI jobs** tab on a book.




You'll notice a couple of other changes in support of Decision Optimization. The Administration perspective now includes an **Integrations** tile, where you can configure the required integration with the IBM Cloud Pak for Data Platform.




Additionally, the properties for an Action button now include a **Run AI Job** tab, where you can specify an existing Watson AI job to run from the button.



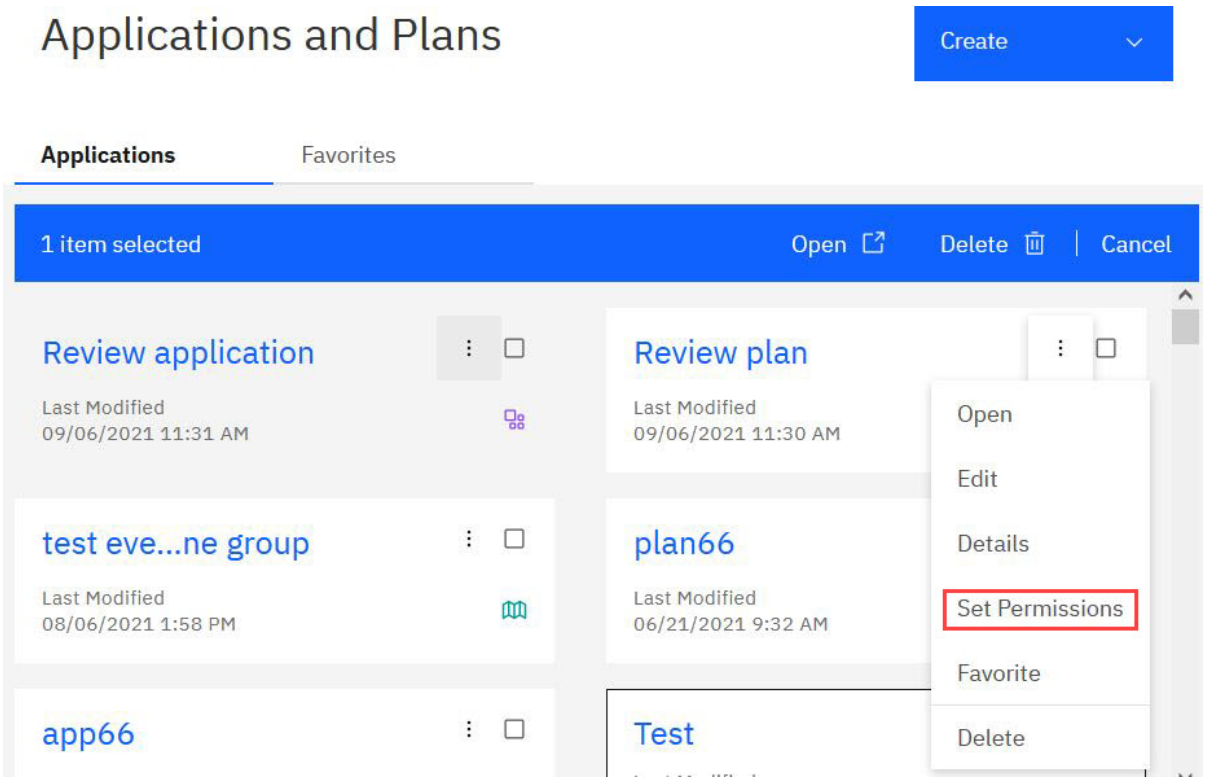

## Analysts and modelers can create applications and plans

Analysts and modelers, not only administrators, can now create and manage applications and plans. The plan or application creator is automatically given full control over the asset.

Any user with the **Full control** permission is considered an owner of the application or plan. Users with **Edit only** or **View only** permissions can contribute to the plan or application when they belong to a group that is added to a step (plan) or section (application).

Users with the **Full control** permission can submit the steps or sections on behalf of any user. They can also give permissions to other users for the assets they own. Users without full control can submit only the steps or sections that they are added to.

To facilitate the process of assigning permissions to other users, users with the **Full control** permission have now the **Set Permissions** option available to them from the plan or application context menu ⋮ in the **Applications and Plans** page.



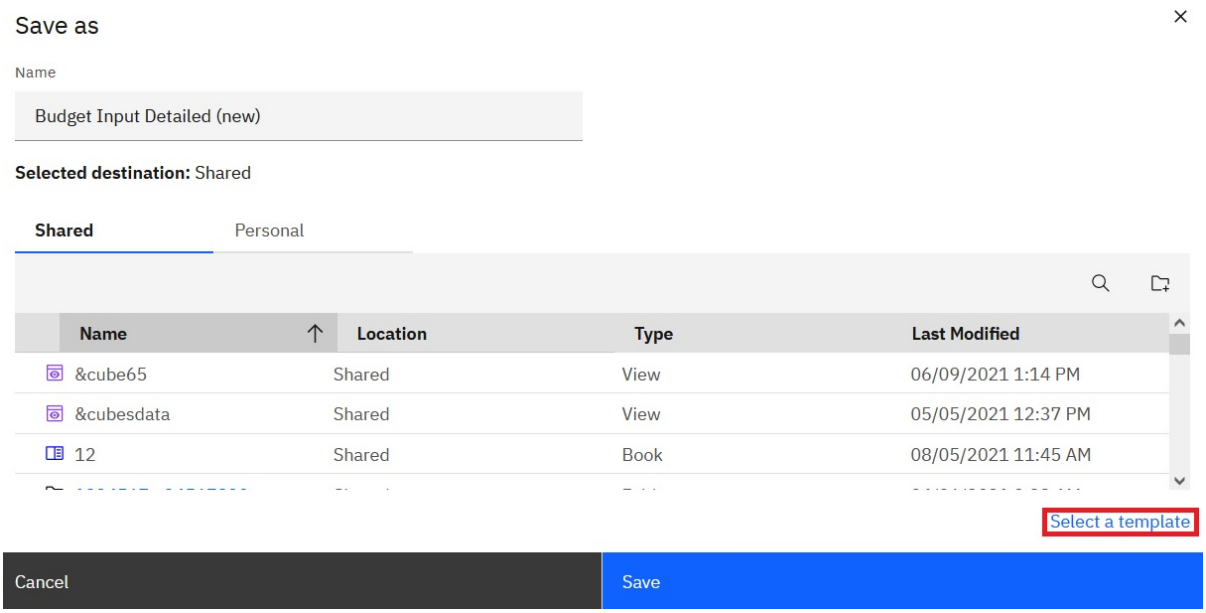

For more information, see Use applications and plans to organize work.

## Dashboard grid size enhancements

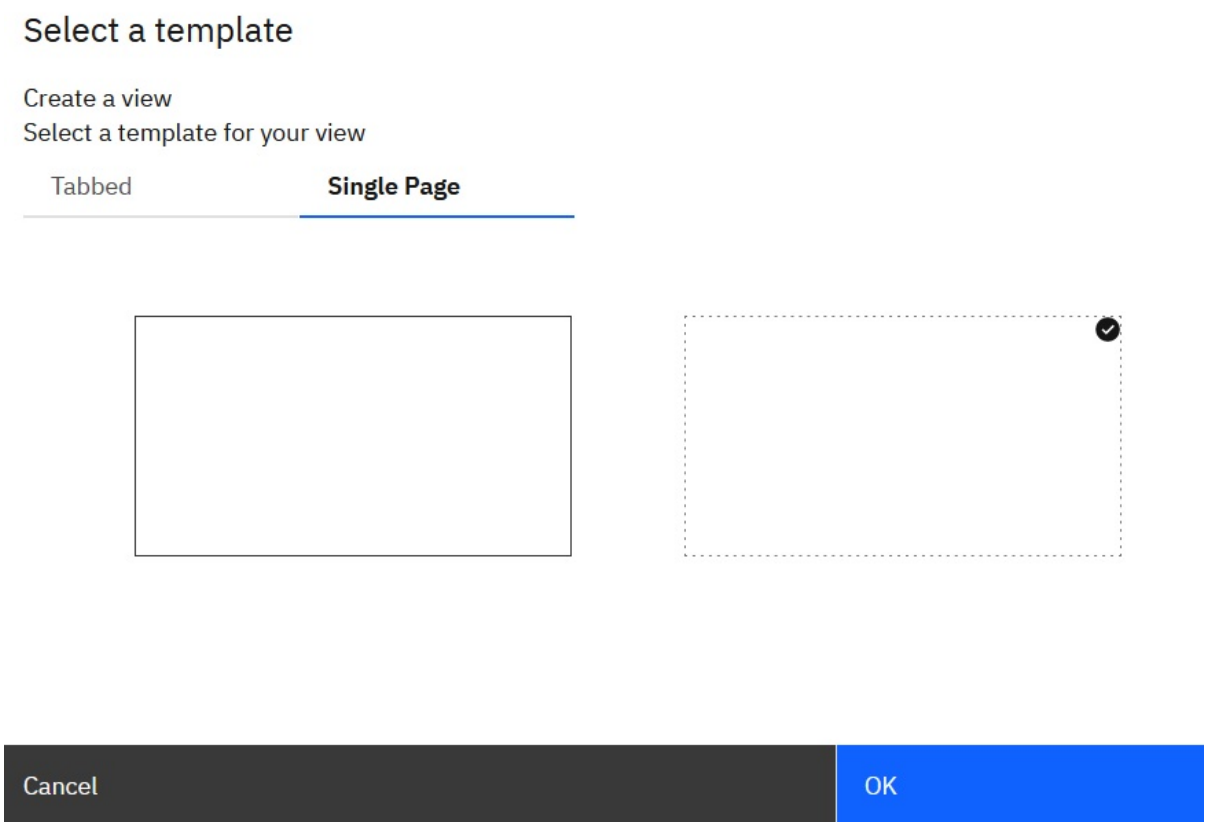
You can now use a template to display widgets in books and websheets in a tabbed page or single page.

In previous versions of IBM Planning Analytics Workspace, a hardcoded template with a single widget was used.

When you save a view by using the **Save view as** option, you can select a template.




When you click **Select a template**, the following dialog box is displayed:




You can save the view as a **Tabbed** or **Single page** page. When the **Tabbed** template is selected, the view is displayed on one tab, and you can add more tabs to the book. When the **Single page** is selected, you cannot add more tabs to the book. With the dotted-line template, the view is maximized to use all space within the tab or sheet. With the solid-line template, the view is displayed with the default widget sizing.

For more information, see Save a view.

### Download a Welcome Kit directly from Planning Analytics Administration

When you receive initial notification that a Planning Analytics Welcome Kit is available, you must download the kit from IBM@Box. However, any subsequent Welcome Kits can now be downloaded directly from Planning Analytics Administration.




For full details, see Download a Welcome Kit from Planning Analytics Administration.

## 2.0.67 - What's new, August 24, 2021

Learn about new features and known issues in version 2.0.67 of IBM Planning Analytics Workspace in the following topics.

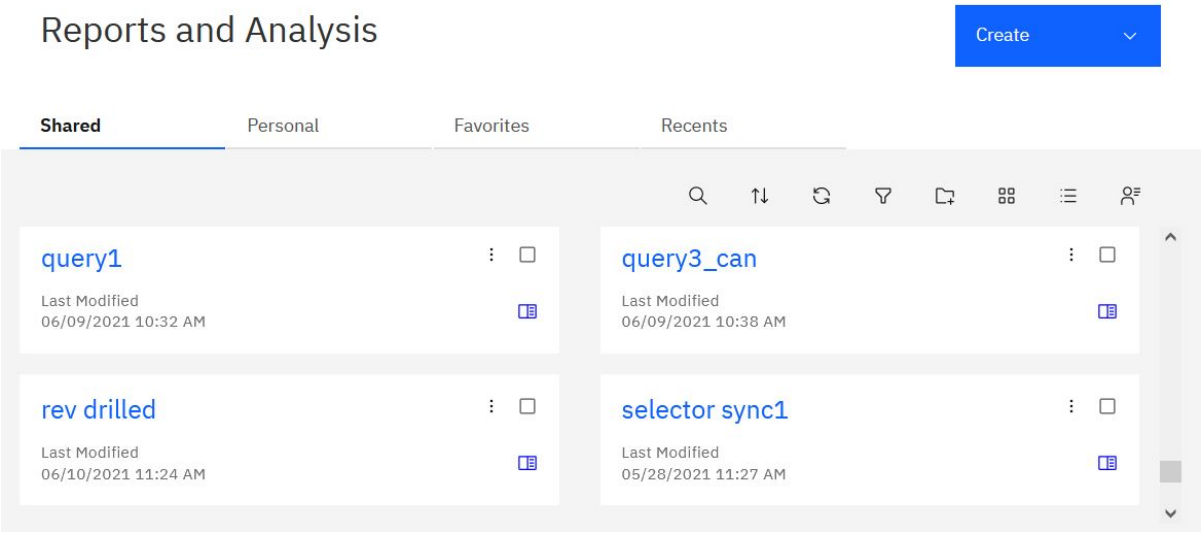
Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

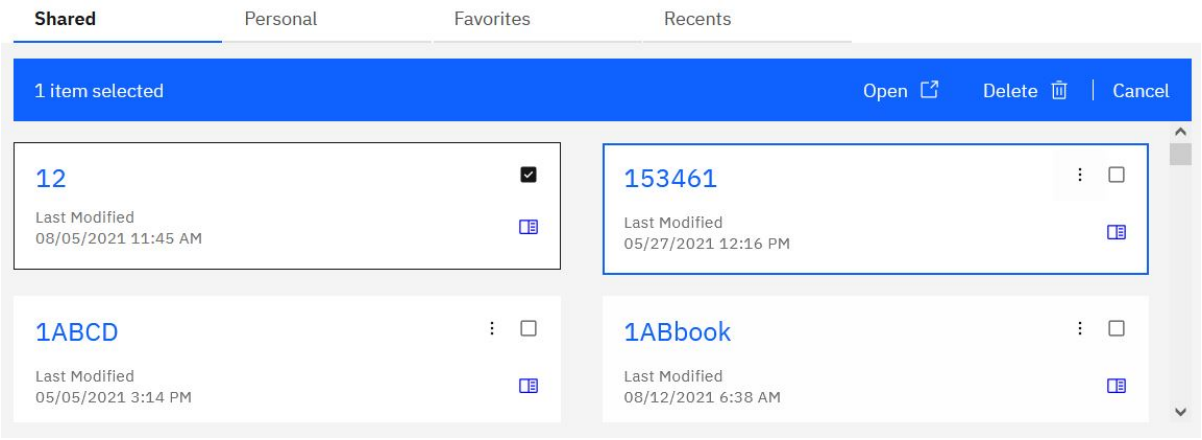
## Checkboxes available in the grid view

The tiles in the grid view in the **Applications and Plans**, **Reports and Analysis**, and **Data and Models** pages now include a checkbox.

The following image shows the **Reports and Analysis** page in the **Grid view** . On each tile, the checkbox is in line with the item name.




When you select the checkbox, the page actions toolbar changes to show options that are available for the selected item or items, as shown in the following image.

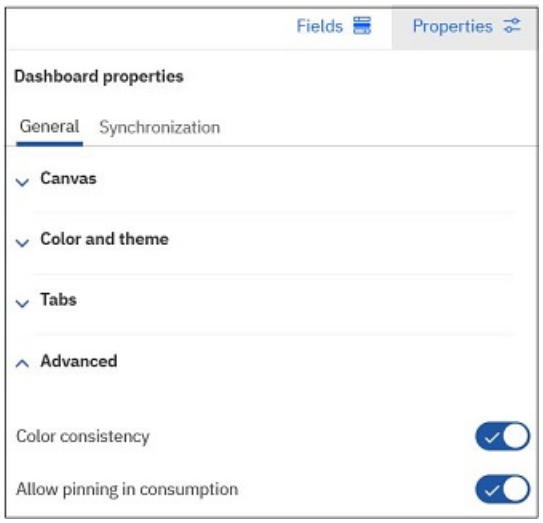



The checkboxes in the **Grid view** and the **List view** are used in the same way.

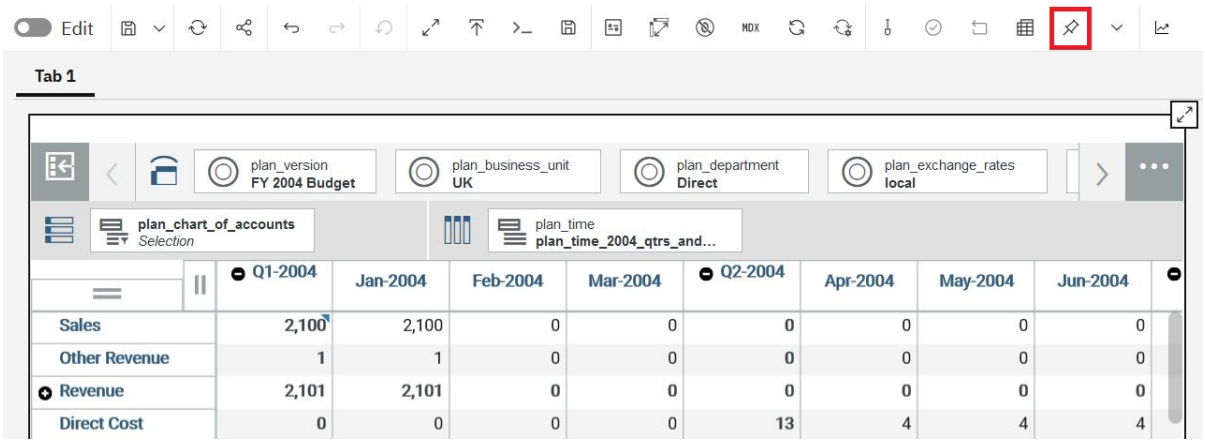
## Pin widgets while in consumption mode

You can pin widgets, such as views and websheets, while in consumption mode.

Until now, the ability to pin a widget in a book was available only in the edit mode. Starting with this release, you can turn on the book property **Allow pinning in consumption** to enable pinning in consumption mode.




The pin button then becomes available in the toolbar when the widget is selected in consumption mode.




Clicking the pin button adds the widget to the **Pins** tab.

For more information, see Enable pinning in consumption mode and Pin items.

## Assign a step or a section to everyone in plans/applications

Until now, you had to assign individual user groups to a step in a plan or to a section in an application. You can now assign a step or a section to Everyone, allowing all users in your environment access to a step or section.

# 2.0.66 - What's new, July 14, 2021

Learn about new features and known issues in version 2.0.66 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

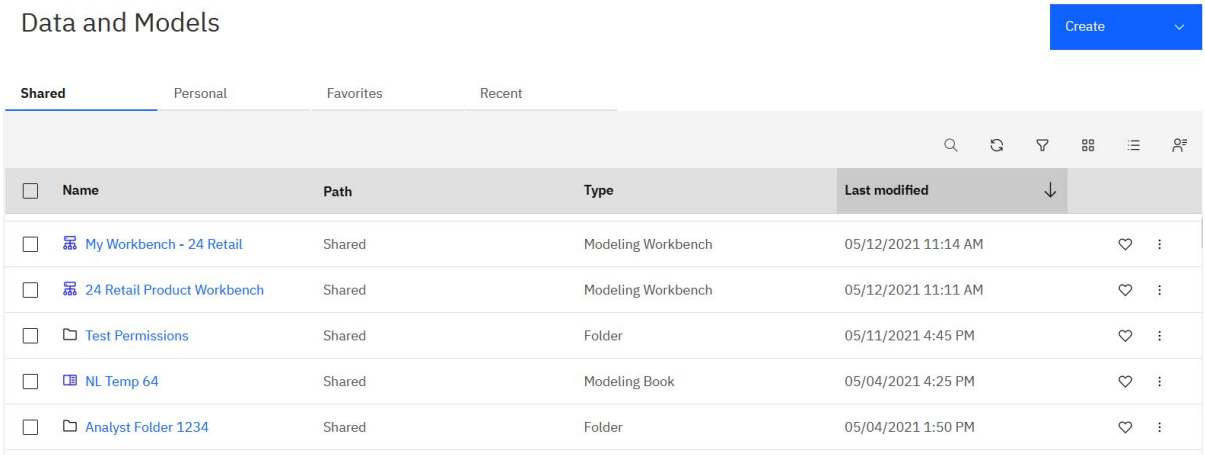
You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

# The modeling workbench is fully integrated with the product

The modeling workbench that was introduced as a beta feature in IBM Planning Analytics Workspace 2.0.64 and 2.0.65 is now available as a standard product feature.

The modeling workbench is used to create and maintain your model. All the workbench functionality is identical as in the beta version. However, with the introduction of the **Data and Models** view in this release, you can access existing workbenches and create new ones in a slightly different way. For example, in previous releases, if you clicked the **Data and Models** tile in the **Quick Launch** section of the Welcome page, the workbench view was opened directly. In this release, this action opens the **Data and Models** view from which you can access existing workbenches and create new ones.

The following screen capture shows the **Data and Models** page where you can access and manage existing workbenches in the **Shared**, **Personal**, or other folders, and use the **Create** button to start creating new workbenches.

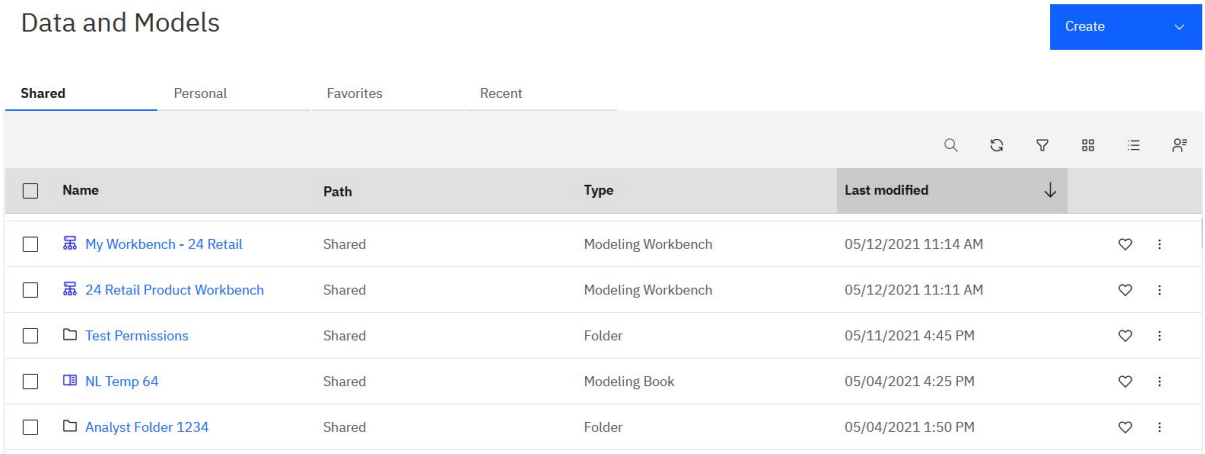



For mu information, see Use the modeling workbench to create and manage your model.

# Full-page view of the modeling content

The **Data and Models** page provides access to the modeling content in IBM Planning Analytics Workspace.

To access the **Data and Models** page, click the **Home** menu ☰, and then click **Data and Models**. You can also access this page from the welcome page, the **Quick Launch** section, the **Data and Models** tile.




From this page, you can create new, and access and manage existing modeling workbenches and modeling books. The modeling content of the **Shared**, **Personal**, **Favorites**, and **Recent** folders is displayed on separate tabs.

The items in each folder can be displayed in the **List view** (default) or **Grid view**.

To open an item, click it. The item is opened in the related user interface. To manage an item, use the options in the actions toolbar or the item context menu.

## Apply proportional or relative proportional data spreading to a forecast

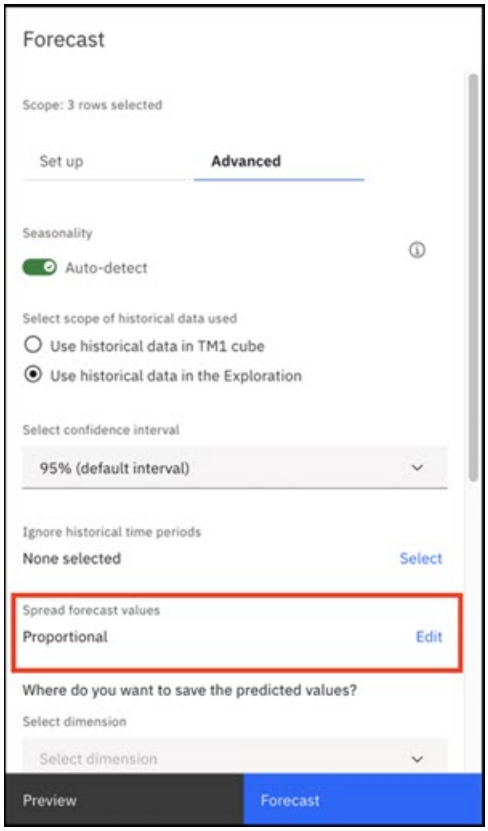
You can now use data spreading methods to write values to consolidated cells when you create a forecast.

**Note:** Data spreading in a forecast requires TM1 server 11.8.7 (PA 2.0.9.8) or later.

Proportional data spreading distributes a specified value among cells proportional to existing cell values.

Relative proportional data spreading distributes values to the leaves (children) of a consolidation proportional to the leaves of a reference cell or cells. This is useful when you want to create a forecast based on a known previous time period.

Both of these spreading methods can be applied to a forecast.




For more information on using data spreading in a forecast, see Forecasting options.

## Hide zero/null data points in select visualizations

You can suppress zeroes in an exploration or visualization, but this only suppresses entire rows or columns that contain zero/null values. Individual zero/null data points cannot be suppressed. Planning Analytics Workspace 2.0.65 introduces a **Hide empty data points** property that lets you hide individual zero/null values in visualizations.

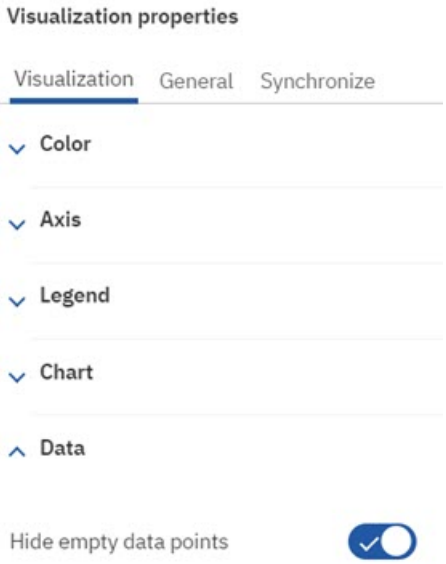
This property is available for the following visualization types:

- Box plot
- Line

- Line & column
- Radar
- Word cloud

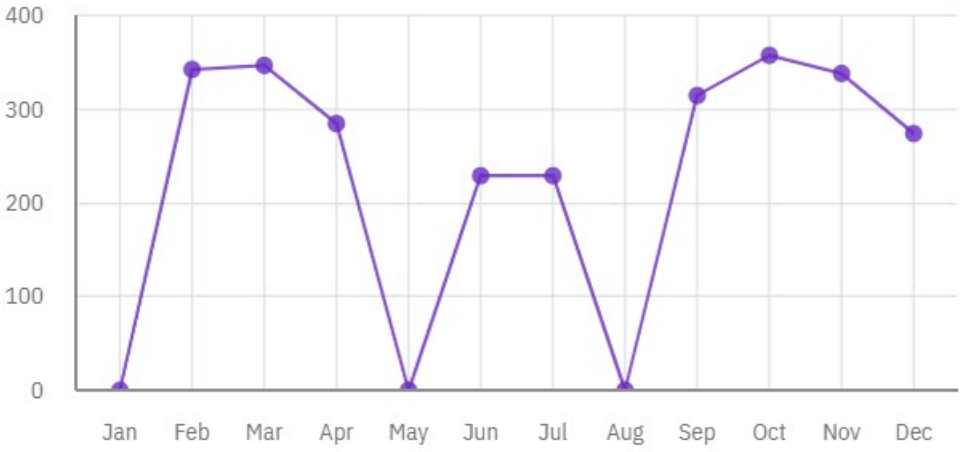
To enable the property:

1. Enable Edit mode on the dashboard.
2. Select a visualization on the dashboard.
3. Click **Properties**.
4. Click the **Visualization** tab.
5. Click the **Data** category.
6. Enable the **Hide empty data points** property.

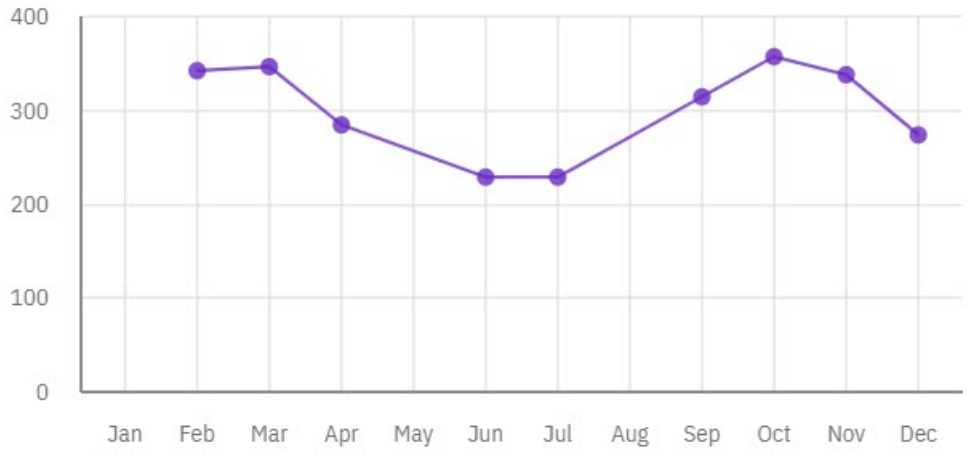



When the property is enabled, empty data points are excluded from your visualization.

For example, here's a simple line visualization for a view that includes several zero values (Jan, May, and Aug). In this image, the **Hide empty data points** property is not enabled. You can see that the zero values are included in the visualization.




When the **Hide empty data points** property *is* enabled, as in the following image of the same visualization, data points for zero values are hidden in the visualization.




## Confirm the Clear spreading shortcut command in the cube viewer

When you type 'c' or 'C' in a cube viewer cell, you are now prompted to confirm that you want to clear values. This ensures that cell values are not inadvertently cleared.

You will be prompted for confirmation when 'c' or 'C' alone is entered in a cell, as well as when entered with trailing or leading spaces or with any of the directional spreading indicators ( | ^ < > ).

## Include attribute headers when exporting a cube view

The cube viewer supports the display of attribute values on both rows and columns. In previous version of Planning Analytics Workspace, attribute values were included when exporting a view to Excel, but attribute names were not exported as a header.

As of Planning Analytics Workspace 2.0.66, attribute names are exported as headers when you export to Excel.

Attributes can exist on rows and columns. When a view contains attributes on **both** rows and columns, there can be a conflict as to which attribute name should show up in the header cell. In this case, the attributes on the rows take priority.

For details on exporting a view, see Export to Excel.

## Use * and ? wildcards to search in the Set editor

You can now use the * and ? wildcard characters when searching in the Set editor. The * wildcard character matches zero or more characters. The ? wildcard character matches any single character.

For further details on searching in the Set editor, see Search in sets.

# 2.0.65 - What's new, June 16, 2021

Learn about new features and known issues in version 2.0.65 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

# Use the modeling workbench to create and maintain your model (beta)

The modeling workbench introduced in IBM Planning Analytics 2.0.64 remains available as a beta feature in 2.0.65.

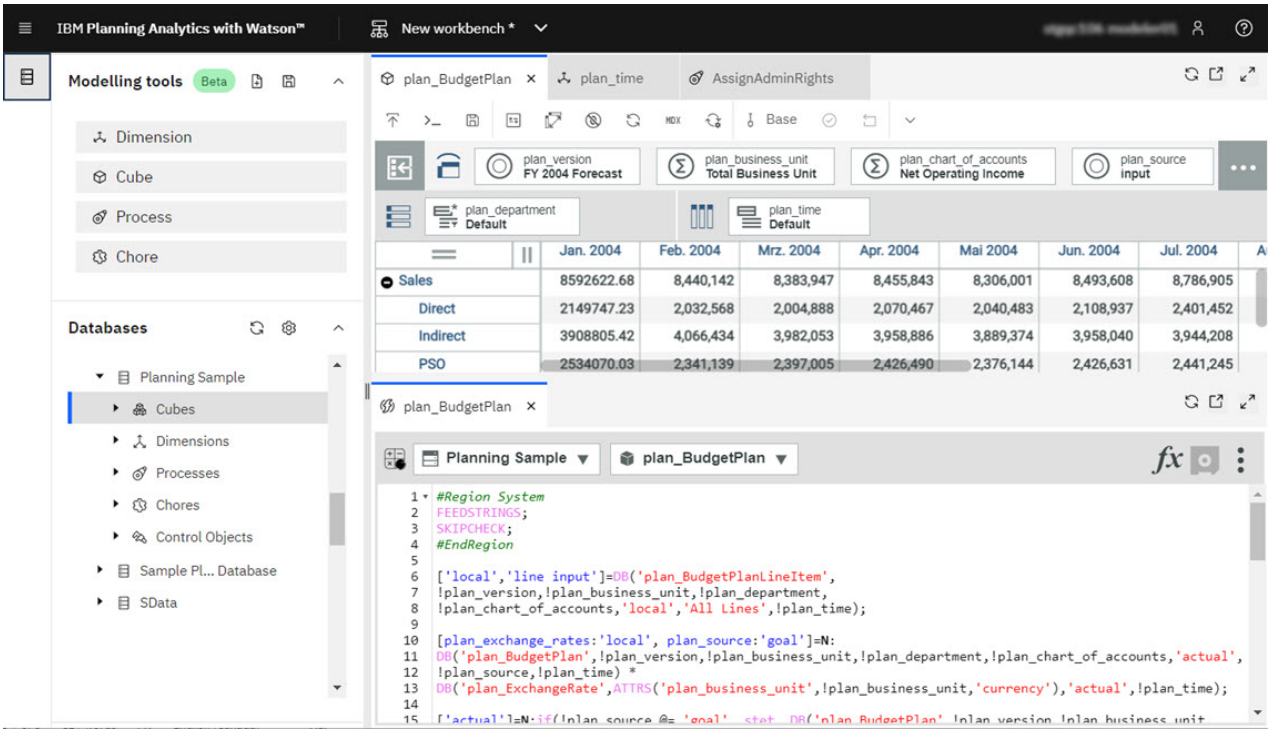
You can now open the modeling workbench from the user interface; you no longer need to enter a URL to the `modeling` perspective in your Planning Analytics Workspace environment.

To create a new workbench, click the **Home** menu ☰, then click **New** > **Workbench**.

To open an existing workbench:

1. Click the **Home** menu ☰, then click either **Shared** or **Personal**, depending upon where the workbench is saved.
2. Click the workbench you want to open. To make it easier to find your workbench, click the **Filter** icon ▽ and select **Workbench**. You can also search for a workbench.

Only users with the modeler or administrator role can create or edit workbenches.




As you can see, the workbench efficiently presents all your modeling assets in a tabbed and sectioned layout.

You can use the modeling workbench to:

- Create, edit, and delete any of these objects: dimensions, hierarchies, sets, processes, chores, and rules
- Run processes and chores
- Create and save cube views
- Edit settings for cubes, dimensions, processes, and chores
- Create drill processes, import data, import dimensions, reload/unload cubes
- Reorder dimensions
- Refresh security, set cell security

For details on using the modeling workbench, see Use the modeling workbench to create and manage your model.

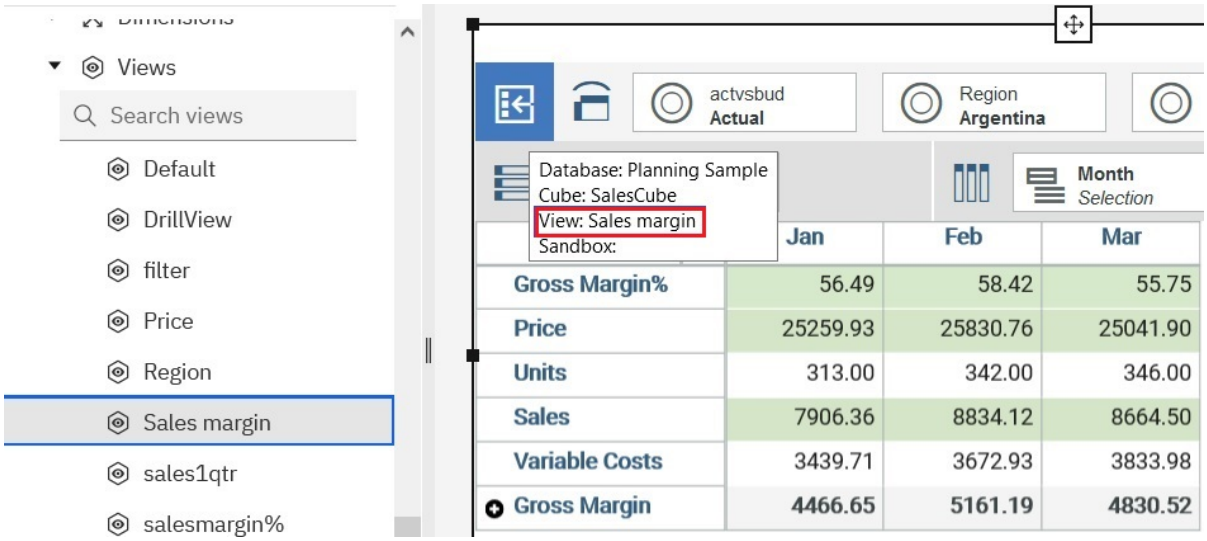
We encourage you to provide feedback on your experience with the modeling workbench during the beta period. Please visit https://community.ibm.com/community/user/businessanalytics/blogs/stuart-king1/2021/05/17/modeling-workbench-beta to share your thoughts.

## See the updated view name when you save a view to the server

When you save a view by using the **Save to server** option, the new view name is displayed in the view details box.

In previous versions of the product, the old view name was displayed here when the view was saved to the server under a different name.

In an exploration, hover the cursor over the view details icon  to reveal information about the view, including the view name.

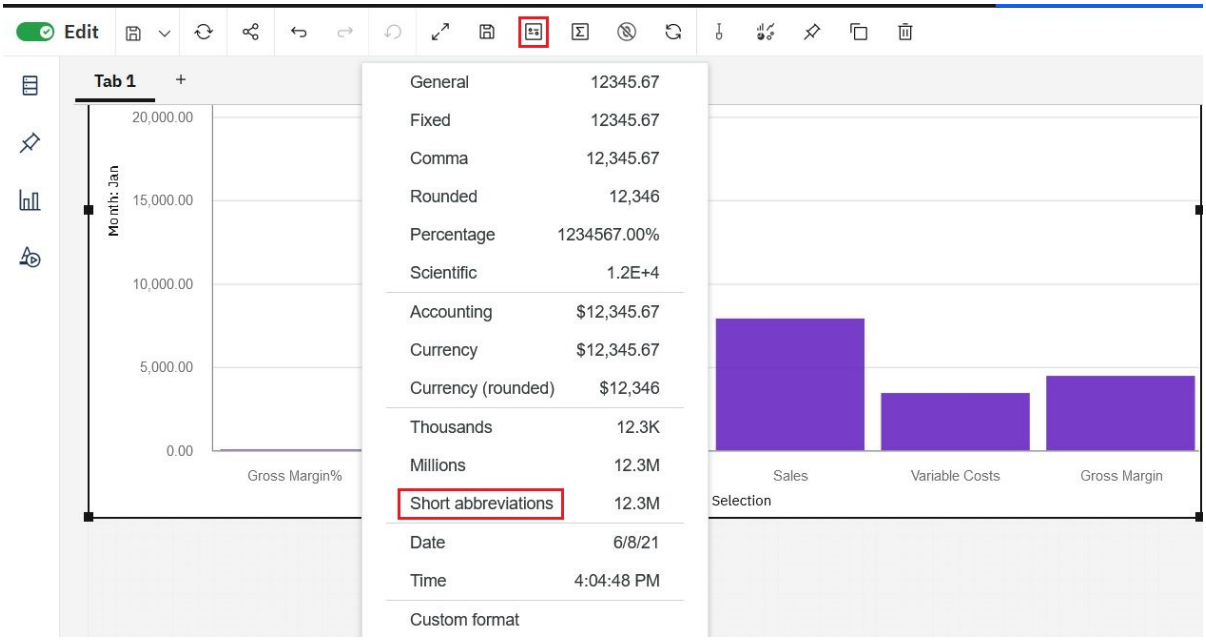



This functionality is available in the Planning Analytics Workspace new experience, and in Planning Analytics for Microsoft Excel.

For more information, see Save a view.

## Abbreviated format to display large numbers

The **Short abbreviations** data format is now available to format large numbers in a view.



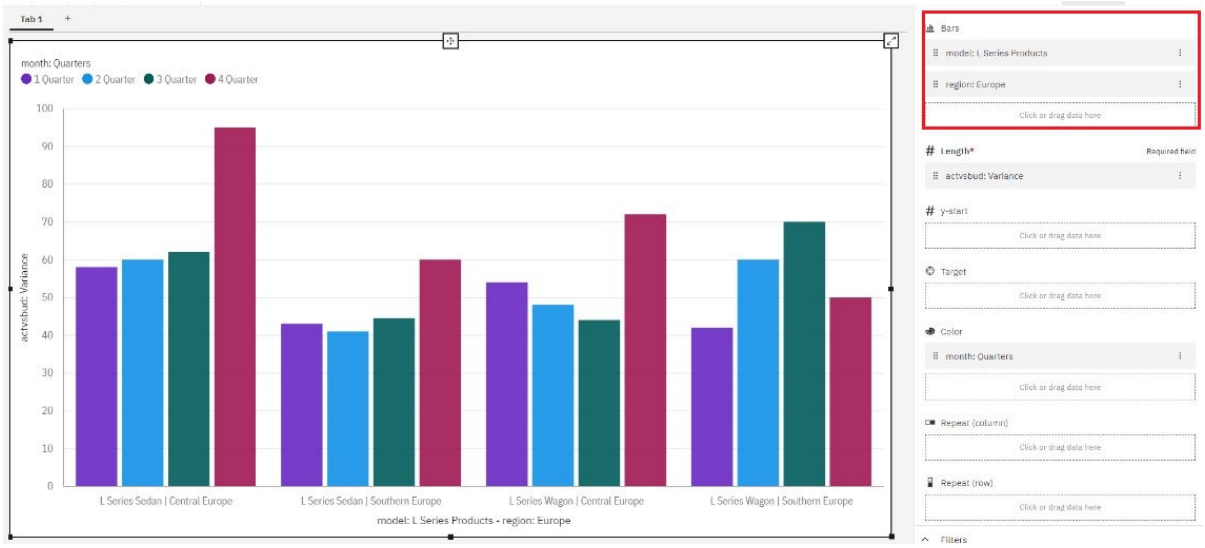

Depending on how large the number is, it's shortened by using a locale-specific abbreviation. In English, it's K for thousands, M for millions, B for billions, and T for trillions. For example, the value of 1500 is shown as 1.5K, and the value of 1500000000 is shown as 1.5B.

For more information, see Change the format of data in a view.

## Assign multiple data items per slot in a visualization

You can assign multiple data items to a single slot in a visualization.

For example, in the following screen capture, the Model and Region dimensions are mapped to the **Bars** field.




In previous versions, only one data item could be assigned to a slot.

## Create calculations for named sets

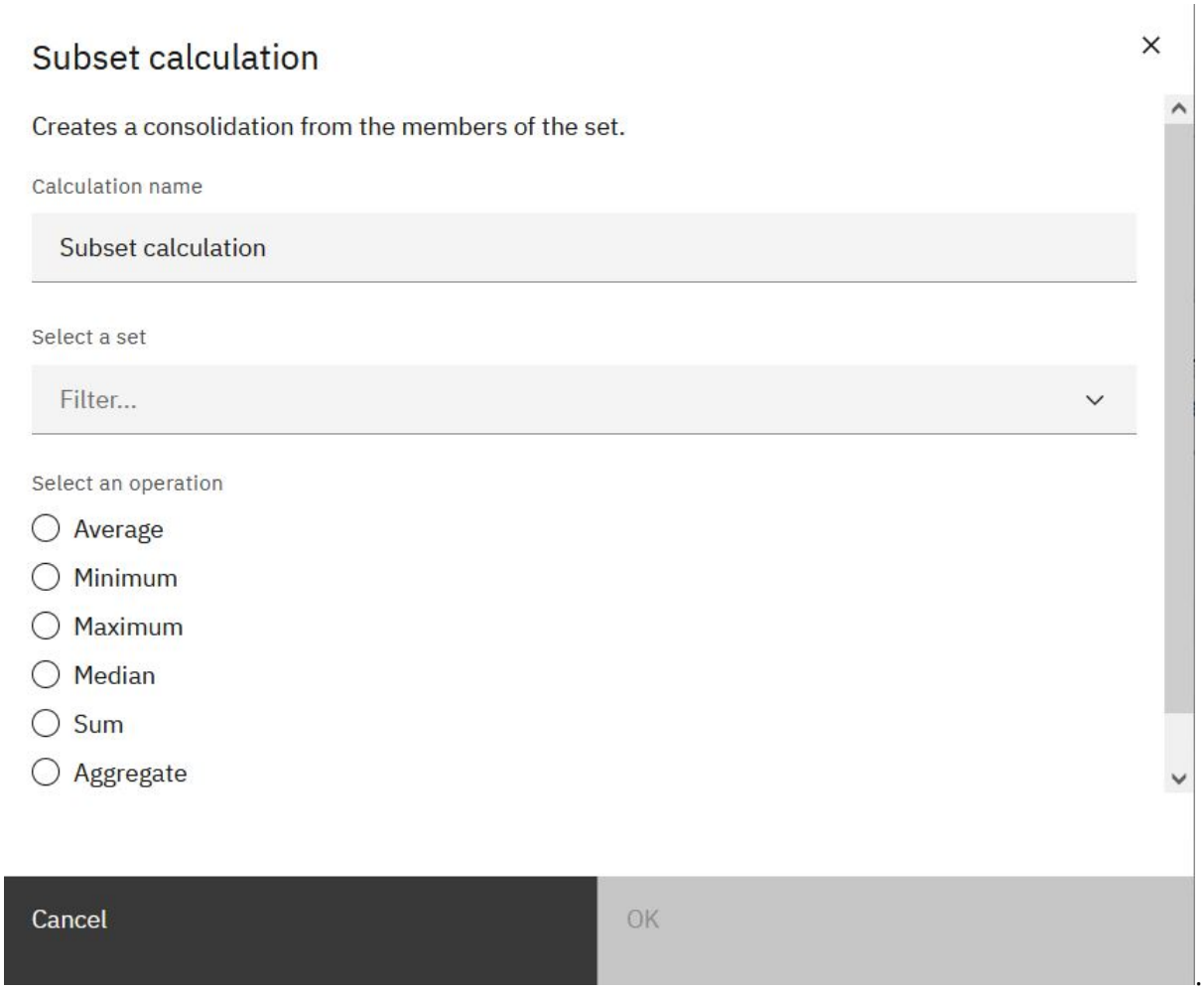
If a dimension includes named sets, you can create a calculation that is based on the set members.

To launch the calculation editor in a view, do the following steps:

1. Right-click a row or column label, or the rows or columns handle.
2. Click **Subset calculation**.

   The calculation options are the same as in the **Summarize all** calculation. The **Aggregate** option is available only on a server that supports aggregates.

   If you created the subset calculation for specific rows or columns, the calculation is added in the grid after the related row or column. If you created the subset calculation from the rows or columns handle, the calculation is added as the last row or column.




For more information, see Add a subset calculation to a view.

## Minor changes to the procedures for creating plans and applications

The user interface and procedures for creating plans and applications have been updated to simplify the creation and editing process.

The capabilities of plans and applications remain unchanged. The information on IBM Documentation describes the new interface and procedures.

# 2.0.64 - What's new, May 18, 2021

Learn about new features and known issues in version 2.0.64 of IBM Planning Analytics Workspace in the following topics.

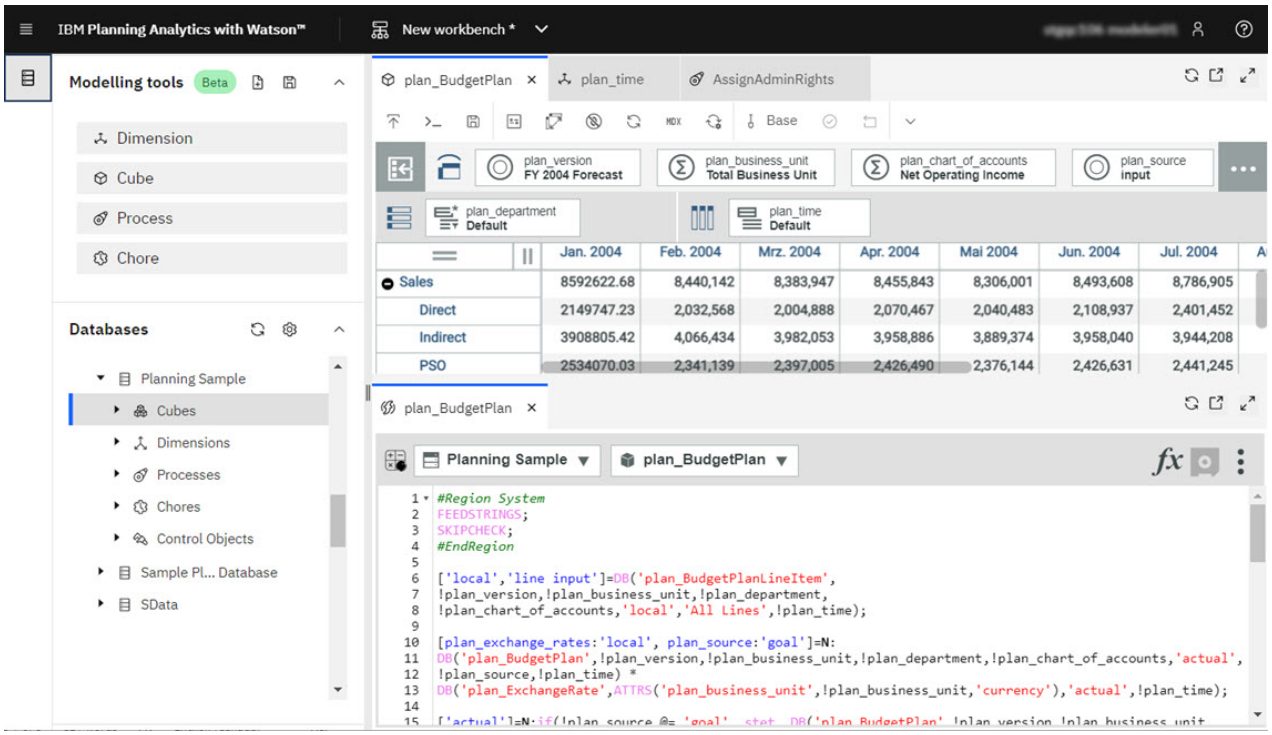
Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Use the Modeling workbench to create and maintain your model (beta)

IBM Planning Analytics 2.0.64 introduces the modeling workbench as a beta feature. The modeling workbench is a dedicated modeling space optimized for all modeling activities.

To use the modeling workbench during the beta period, open the `modeling` perspective in your Planning Analytics Workspace environment. For example, `https://<my_planning_analytics_url>/?perspective=modeling`. Only users with the modeler or administrator role can create or edit workbenches.




As you can see, the workbench efficiently presents all your modeling assets in a tabbed and sectioned layout.

You can use the modeling workbench to:

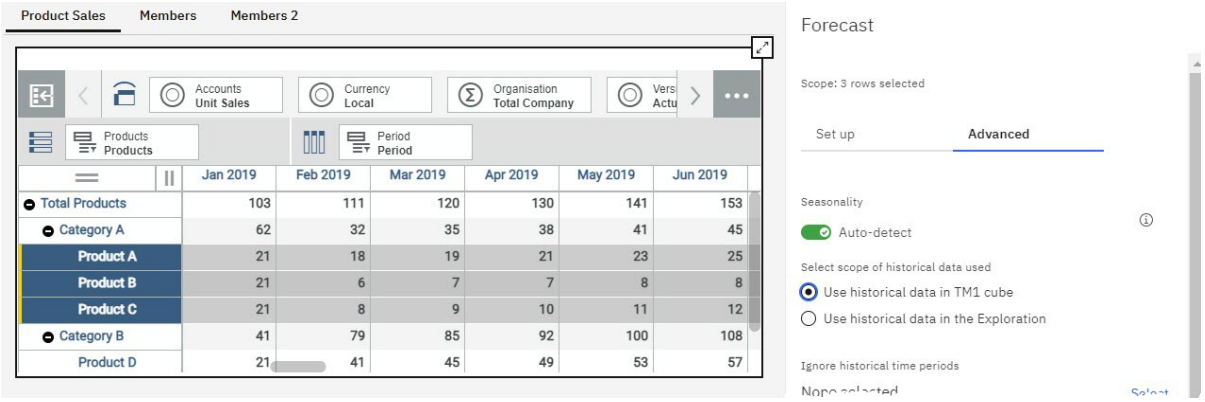
- Create, edit, and delete any of these objects: dimensions, hierarchies, sets, processes, chores, and rules
- Run processes and chores
- Create and save cube views
- Edit settings for cubes, dimensions, processes, and chores
- Create drill processes, import data, import dimensions, reload/unload cubes
- Reorder dimensions
- Refresh security, set cell security

For details on using the modeling workbench, see Use the modeling workbench to create and manage your model.

We encourage you to provide feedback on your experience with the modeling workbench during the beta period. Please visit https://community.ibm.com/community/user/businessanalytics/blogs/stuart-king1/2021/05/17/modeling-workbench-beta to share your thoughts.

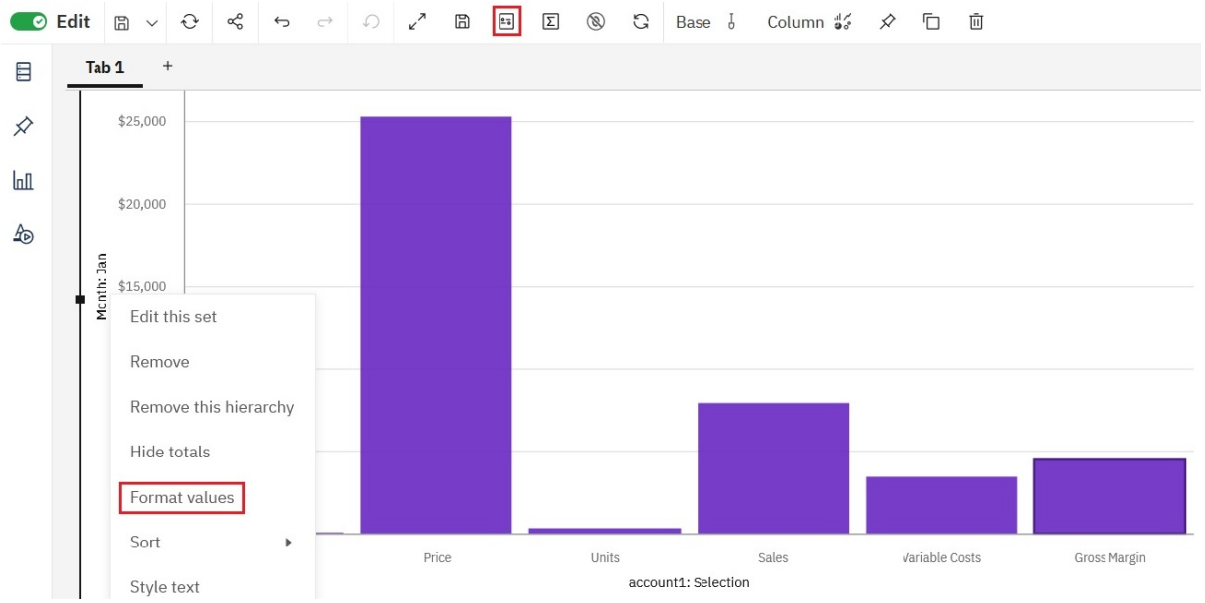
## Forecasting improvements

As of Planning Analytics Workspace 2.0.64 you can define historical multiple time periods to be ignored by the forecasting engine.




## Apply value formats in visualizations

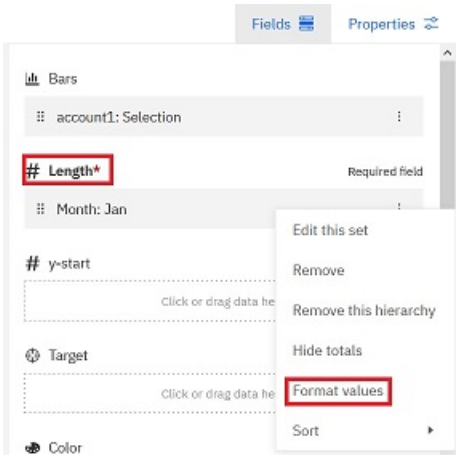
You can now format data values in visualizations. The formats are retained when the view is switched back to exploration or other type of visualization.




The **Format values** icon in the application toolbar is now available for visualizations. Use this icon to set data format for all values in the view.

To set data format for rows or columns, right-click the column in the visualization, and click **Format values**.

You can also set the format in the **Fields** tab.

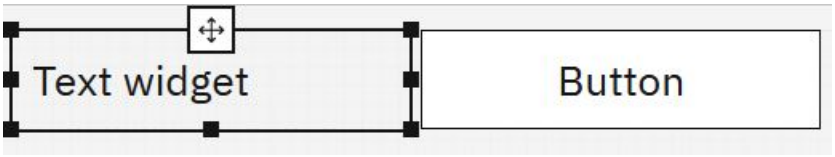



This functionality is available only in the Planning Analytics Workspace new experience.

For more information, see Change the format of data in a view.

## Identical font size in equally sized widgets

When equally sized text, cell, or button widgets are placed in a book, the font size of the text in each widget is identical.




This was not the case in previous releases, where the text font size in a cell widget was much smaller than the content in an equally sized text widget.

For more information, see Add cell values to the sheet as a cell view.

# 2.0.63 - What's new, April 21, 2021

Learn about new features and known issues in version 2.0.63 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

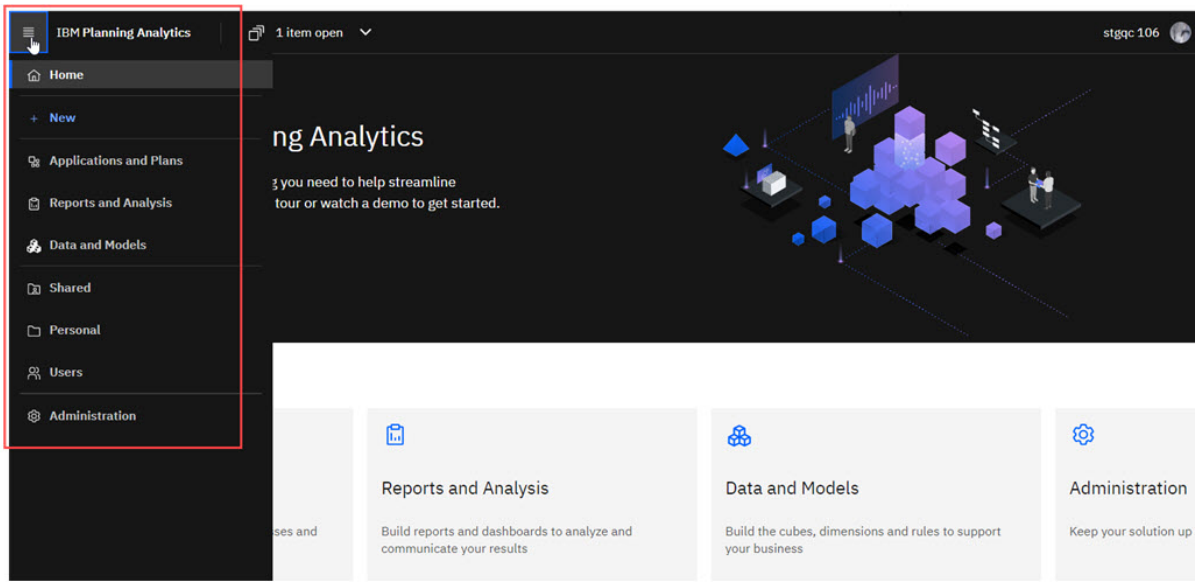
You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Design and usability improvements

Several design and user interface enhancements are included in the Planning Analytics Workspace 2.0.63 release, signifying our continued investment in usability. Here are a few of the areas we have improved in Planning Analytics Workspace 2.0.63.

### Collapsible navigation panel

The navigation panel is now available from the Home menu.

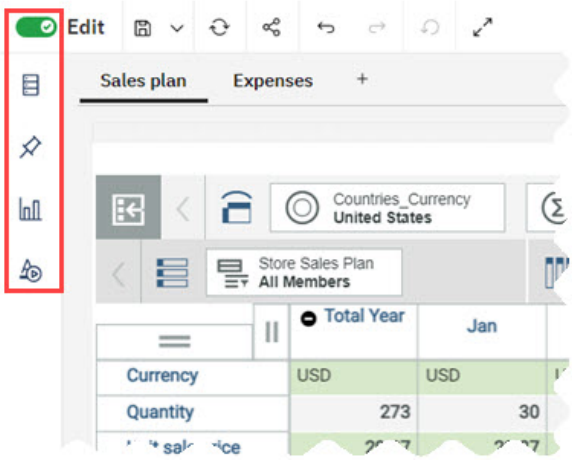



You no longer need to return to the Workspace home page to navigate or access Workspace content. For example, if you're working on the Administration page, you can navigate directly to a book by opening the navigation panel, opening the Shared folder, and searching for the book.

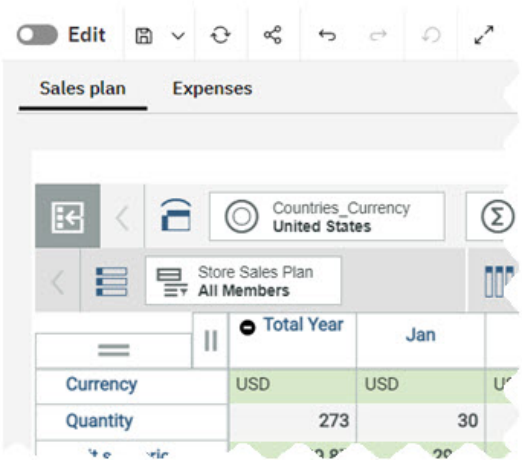
## Database Tree hidden when not in edit mode

The database tree is hidden when a book is not in edit mode. To reveal the database tree, the book must be set to edit mode.

Edit mode enabled




Edit mode disabled

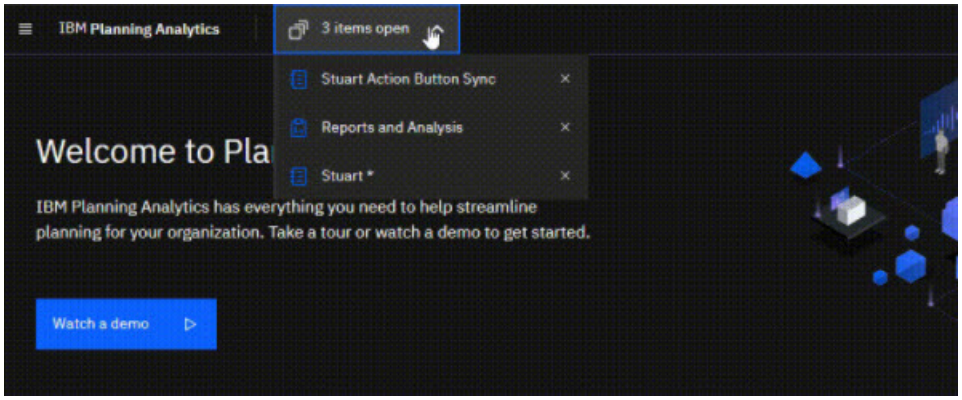



This change provides more horizontal space for users consuming the book. This change also makes it clear that you must enable edit mode to incorporate content from the database tree into the book.

Note that the Edit pencil icon has been replaced with a labeled Edit switch. This change makes it easier to discern when a book is in edit mode.

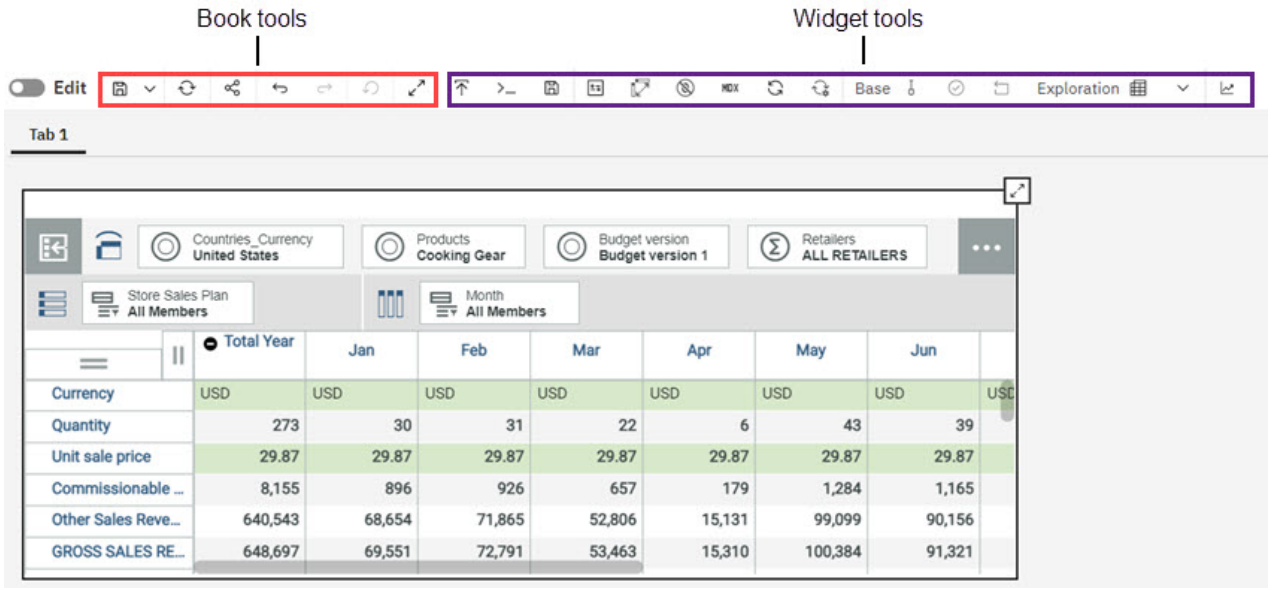
## Left-aligned switcher

The content switcher is now left-aligned for a consistent look and feel with Cognos® Analytics. The content switcher no longer includes a Home option. To return to the home page, click **IBM Planning Analytics**.



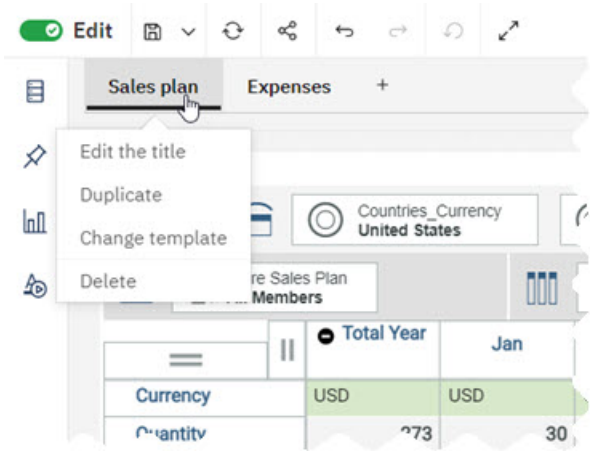

## Combined toolbar

Book and widget toolbars are now integrated into a single toolbar. This change improves the vertical space available to book creators and consumers. Buttons for each widget are displayed alongside the book buttons after selecting the widget.




## Tab controls

Clicking on a tab while in edit mode now presents a menu of available actions.



### Save and Move dialogs

The Save and Move dialogs have been updated to support searching, sorting, creating folders, and replacing books. You can search for folders, books, and views in either the shared or personal folder.

## Rename a TM1 database

Planning Analytics on Cloud customers can now rename TM1 databases by using Planning Analytics Workspace **Administration**.

The self-service updates the database directory in the `prod` folder, and the database name in the `tm1s.cfg` file to match the new name.

You must use the Planning Analytics Workspace new interface, which is also referred to as New Experience, to rename the database. This functionality is not available in the Planning Analytics Workspace Classic Experience.

For more information, see Rename a database.

## Remove users in Planning Analytics Workspace

Planning Analytics on Cloud administrators can remove multiple users at once in Planning Analytics Workspace **Administration**.

Administrators remove users only from the primary tenant (environment), and the system then removes the users from all secondary tenants (environments). The users are automatically removed from IBM Subscriptions Management, from groups that they were members of, and from all environments that they were associated with. Each user's personal folder is renamed to **Unknown**, and the user's personal assets are moved to this folder. User's shared assets remain in the shared folder.

For more information, see Remove users in Planning Analytics Workspace.

## Use the new Box Plot visualization

Use a box plot visualizations to identify outliers and compare distributions in your data.

You can create a box plot to show the median, quartiles, and outlier and extreme values for a variable. The inter-quartile range is the difference between the 75th and 25th percentiles and corresponds to the length of the box. The middle line is the 50th percentile.

Above and under each box, 'whiskers' give additional information about the spread of the data.

Outliers are represented by "o" signs beyond the whiskers.

The mean score in a box plot is presented by a "+" sign.

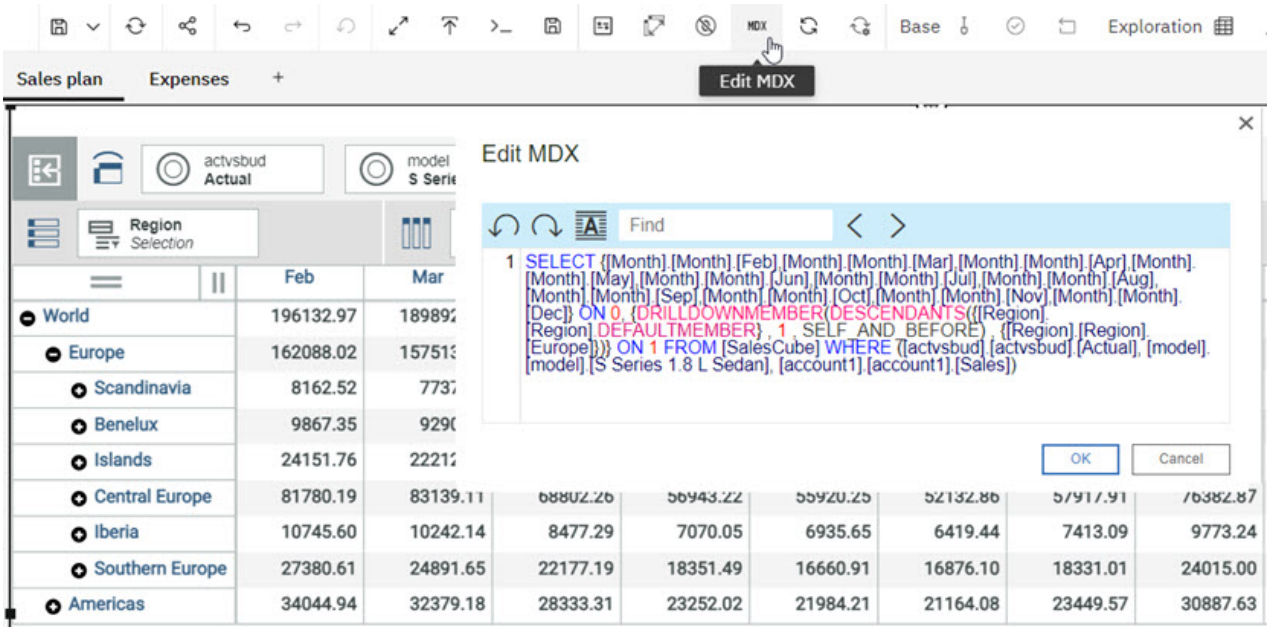
Use the **Key** field in a box plot visualization to determine the items for which you want to identify outliers and compare distributions.

This box plot example shows the gross profit statistics for various markets.




## Edit MDX source for a cube view

You can use the new **MDX** button on the toolbar to view and edit the MDX source for a cube view.




# 2.0.62 - What's new, March 17, 2021

Learn about new features and known issues in version 2.0.62 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Delete a TM1 database

Planning Analytics on Cloud customers can now delete TM1 databases by using Planning Analytics Workspace **Administration**. You no longer need to submit a support ticket to have a TM1 database deleted for you.

The self-service delete action removes the database from the user interface. However, the database directory and its contents are retained on the data tier. You can later decide whether they should be deleted, moved, copied, or kept on the data tier.

You must use the Planning Analytics Workspace new interface, which is also referred to as New Experience, to delete the database. This functionality is not available in the Planning Analytics Workspace Classic Experience.

**Note:** To rename a TM1 database, you still need to open a support ticket.

For more information, see Delete a database.

## Seamlessly scroll through all objects in the data tree

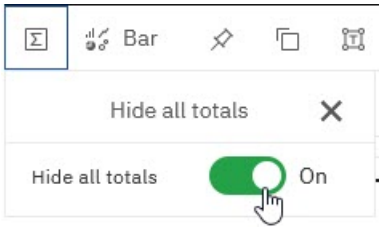
The **Load more** setting has been removed from the data tree in Planning Analytics Workspace. You can now scroll infinitely through all items in a node in the data tree.

The **Load more** setting for the data tree was introduced in Planning Analytics Workspace 2.0.61. This setting determined the number of items within a node that could be viewed before you had to click **Load more** to retrieve and display additional items. With the introduction of infinite scrolling, this setting is no longer necessary and it has been removed from the Settings options.

## Hide totals in a visualization with a new toolbar button

You can now hide totals in visualizations with a couple of clicks on a new toolbar button.

To hide totals in a visualization, click the new **Hide all totals** button on the toolbar, then turn on the **Hide all totals** option.




To restore totals in the visualization, turn off the **Hide all totals** option.

## Retrieve data from multiple TM1 databases in a websheet (Planning Analytics Workspace local only)

You can now retrieve data from more than one TM1 database in a websheet.

All databases referenced in a websheet must be registered on the same Admin host. You can retrieve data from multiple databases only in Custom Reports and Dynamic Reports, which use the VIEW function to retrieve data.

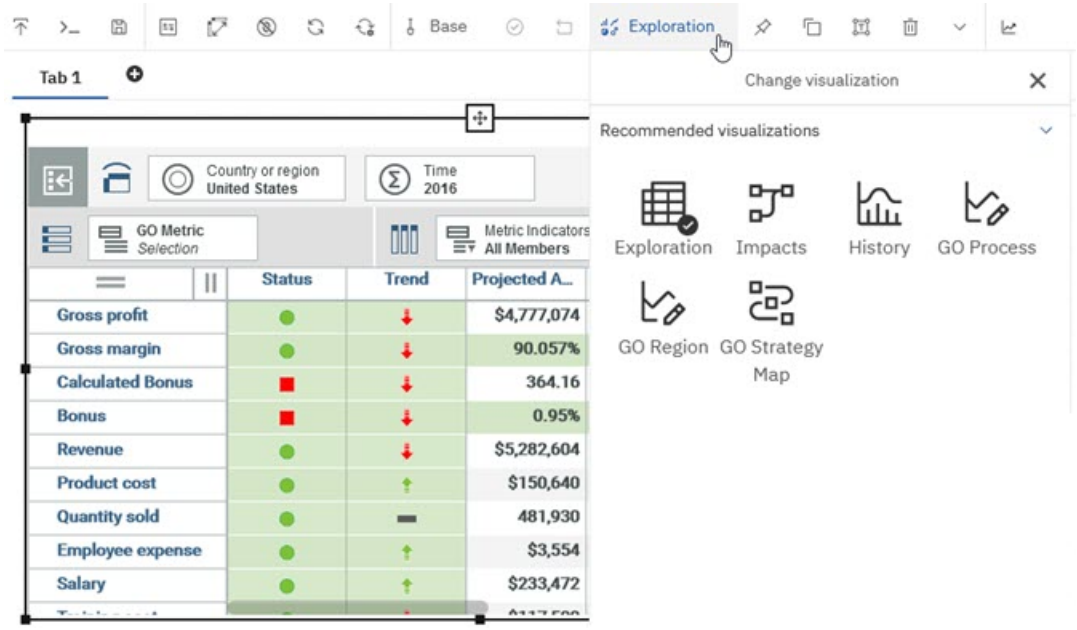
To successfully reference multiple databases in a websheet:

- Database names must be static; computed database names are not supported.
- All databases must be configured to use the same authentication mode.
- Users must have common credentials across databases.

You can learn more about websheets in the Websheets overview.

## Use scorecards to track objectives

If a view is configured to support scorecarding, you can now select from five scorecards to view data and track the progress of your organization.




To display a scorecard, click the **Change visualization** button and pick one of the following scorecards types:

**Impacts**
Illustrates the positive and negative relationships between the metrics in your metrics cube. This type of diagram shows how the business works by displaying how one metric influences another.

**History**
Shows a column chart of data for a metric. By default, it compares the actual value against the target value for each time period, and indicates whether the result is within an accepted tolerance.

**Process**
Shows metrics in the context of a process flow.

**Region**
Shows performance across geographies on a map overlay.

**Strategy Map**
Tracks business performance by perspectives, objectives, and metrics.

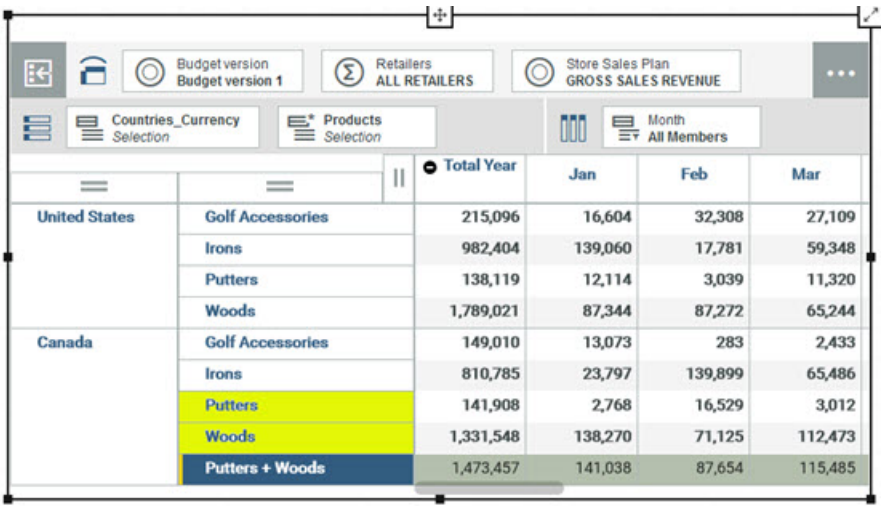
To learn more about scorecards, see Explore scorecards.

## Create asymmetric calculations on stacked rows or columns in an exploration

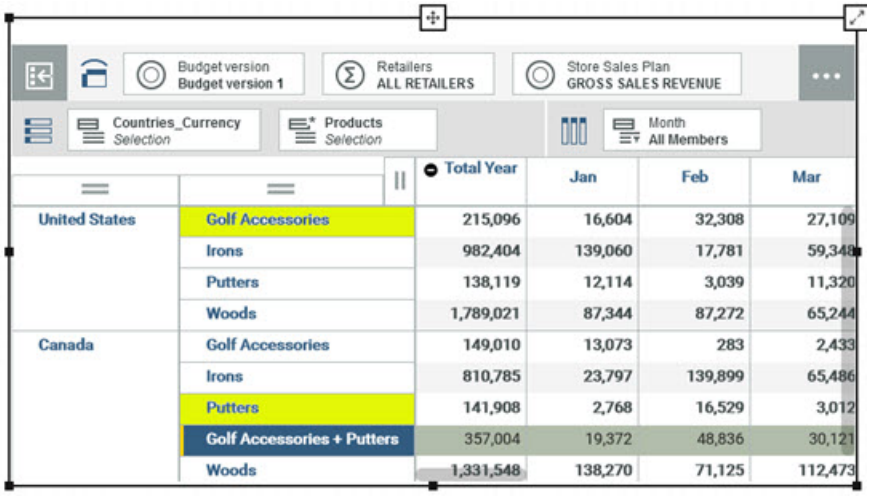
You can now create calculations that apply to a single nested dimension set on rows or columns in a view. Because these calculations are applied to a single nested set for a given dimension and are *not* common to all nested sets for the dimension, they are called *asymmetric calculations*.

Asymmetric calculations can be confined within a single nested set or span multiple nested sets of the same dimension.

In this example, which is confined within a single nested set, you can see the **Putters+Woods** asymmetric calculation in the Products set. It calculates the total of Putters and Woods in Canada. Note that the calculation appears just once; it is not duplicated in both nested Products sets.




This example shows the **Golf Accessories + Putters** asymmetric calculation, which spans multiple nested sets of the Product dimension. It calculates the total of Golf Accessories in the United States plus Putters in Canada. Though the calculation spans the nested Products set for both United States and Canada, it appears only in the nested Products sets for Canada.

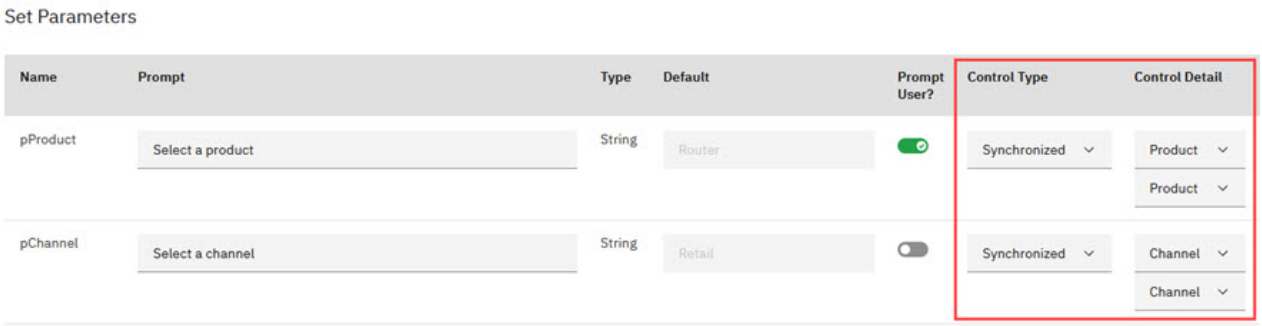



For full details, see Add an asymmetric calculation to a view.

## Synchronize parameters for an Action button that runs a TurboIntegrator process

You can now synchronize string parameters for a TuroIntegrator process that is executed from an Action button, so the process picks up parameter values from other synchronized objects in a book.

When setting a parameter for a process that is executed from an Action button, select the **Synchronized** Control Type and select the dimension and hierarchy to be synchronized under the Control Details column.




When a parameter is set to **Synchronized** and the **Prompt User?** option is enabled, the user is presented with an informational screen indicating the parameter values being used, but cannot change the parameter values.

A process can receive synchronized parameter values from any of the following items:

- cube view (exploration or visualization)
- websheet
- dimension selector
- synchronization tile

## Work with values formatted as Percentage in the Cube Viewer

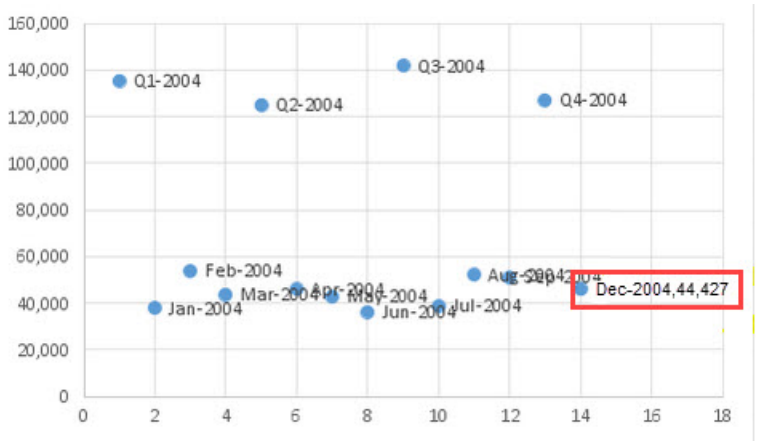
When you double-click a cell formatted as Percentage in the Cube Viewer, you now continue to see the value expressed as a percentage. In previous versions of Planning Analytics Workspace, double-clicking on a cell formatted as Percentage would display the underlying decimal value as stored in the Planning Analytics database.

Additionally, when you modify a value in a cell formatted as Percentage, you now always see the value you entered expressed as a percentage. In previous versions of Planning Analytics Workspace, when you entered a value in a cell formatted as Percentage while the **Defer on leaf data change** option was enabled, the value was displayed as decimal, and would be updated to percentage only after refreshing the view.

## Known issue - Unexpected label in websheet Scatter Chart

This issue occurs when label text in a Scatter Chart is edited to use customized/hardcoded text.

Instead of displaying the customized label text, the label displays "[x-value], [y-value]". X value and y value are the values for the data point as set in the range of the chart.The highlighted label in this image illustrates the issue.




This issue will be corrected in an upcoming release.

# 2.0.61 - What's new, February 10, 2021

Learn about new features and known issues in version 2.0.61 of IBM Planning Analytics Workspace in the following topics.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

You can view the fix lists for this and previous versions of Planning Analytics Workspace at https://www.ibm.com/support/pages/ibm-planning-analytics-20-fix-lists#anchor2.

## Create a TM1 database

Planning Analytics on Cloud customers can now create their own TM1 databases by using Planning Analytics Workspace **Administration**. You no longer need to submit a support ticket to have a TM1 database created for you.

You must use the Planning Analytics Workspace new interface, which is also referred to as New Experience, to create the database. You can even create a database while using the New Experience in Preview mode; there's no need to commit to a permanent upgrade! You can learn more about enabling the New Experience in Preview mode in this Knowledge Center topic: Manage the new experience for your users.

You cannot create a database in Planning Analytics on Cloud using the Planning Analytics Workspace Classic interface.

The database is created in the standard location. You can configure and manage the database just like any existing TM1 database. All users with the Planning Analytics Workspace Administrator role are given ADMIN access to the new database.

To rename or delete the TM1 database that you created, you still need to open a support ticket.

For more information, see Create a database.

## Welcome Kit change for Planning Analytics on Cloud

The security mode 1 "admin" user name and password for each TM1 database are no longer included in the Welcome Kit for Planning Analytics on Cloud.

This change reduces the size of the Welcome Kit and eliminates the need to regenerate and distribute the kit after each request to create, rename, or delete a TM1 database.

When a database is created, a randomly generated password is set for the security mode 1 "admin" user name. If you need to use the admin user, administrators can set a new password by using the AssignClientPassword TM1 function.

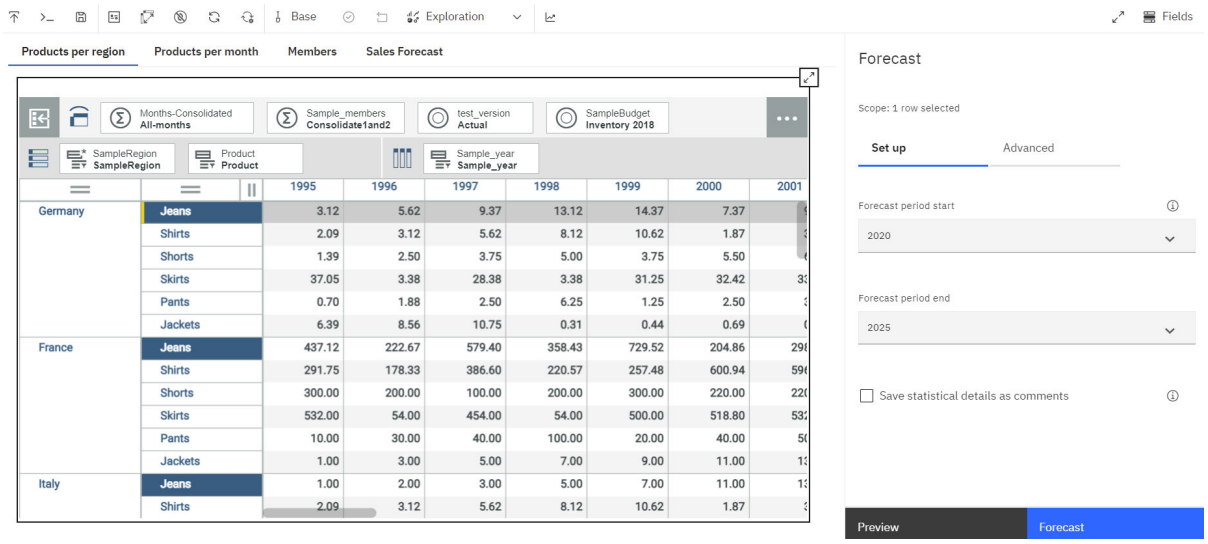
For more information, see The Welcome Kit.

## Forecasting: preview nested dimension

Forecasting in IBM Planning Analytics Workspace allows previewing of nested members. And you can forecast by using cube view data.

**Preview nested dimensions**

As of SC61 Planning Analytics Workspace supports the preview of nested dimensions. Planning Analytics Workspace can now render a preview with nested dimensions. In the following case, jeans is selected for preview and **Preview** is enabled.




For complete details on using forecasting in Planning Analytics Workspace, including a complete tutorial, see Forecasting.

## Sixty minute inactivity timeout for Planning Analytics Workspace on Cloud

A new inactivity timeout enhances security in Planning Analytics Workspace 2.0.61.

Currently, Planning Analytics Workspace on Cloud enforces a 24 hour session timeout. This session timeout disconnects a user 24 hours after their most recent login and requires the user to log back in to Planning Analytics Workspace. The session timeout is enforced regardless of user activity. If a user is active continuously, the session timeout is still triggered 24 hours after the most recent login.

In addition to this session timeout, Planning Analytics Workspace 2.0.61 introduces an inactivity timeout. When a user is inactive for 60 minutes, the user is logged out of Planning Analytics Workspace. This inactivity timeout is mandatory and applicable to all Planning Analytics Workspace on Cloud customers. The 60 minute inactivity interval is static and cannot be modified.

Planning Analytics Workspace will warn a user about an imminent timeout due to inactivity. If the user does not intervene, the inactivity timeout occurs 60 minutes after the most recent activity.

When an inactivity timeout does occur, what is the impact on the user?

- Workspace end user state will be lost.
- Authoring of Books and Plans/Applications that are not saved will be lost.

This new inactivity timeout is applicable only to Planning Analytics Workspace on Cloud, but also impacts Planning Analytics for Microsoft Excel.

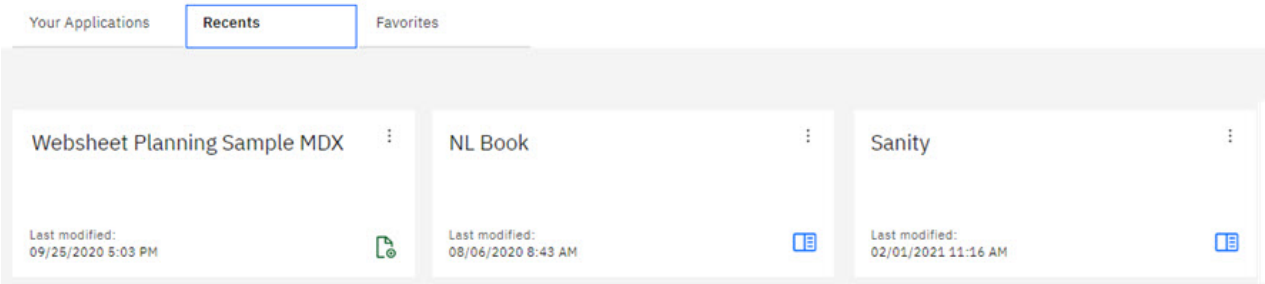
For more information on the inactivity timeout, see Connection timeouts in Planning Analytics Workspace on Cloud.

For details on configuring timeouts in Planning Analytics Workspace Local, see Configuring parameters for Planning Analytics Workspace Local.

For more information on how the inactivity timeout impacts Planning Analytics for Microsoft Excel, see Timeouts in IBM Planning Analytics for Microsoft Excel

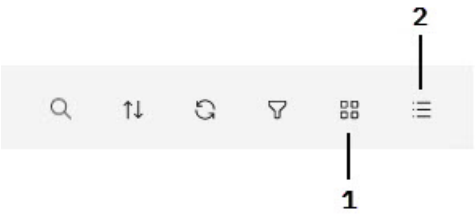
## View assets as tiles

**Recents** and **Favorites** on the Home page are now displayed as tiles.




You also now have the option to view assets on the **Applications and Plans** page and the **Reports and Analysis** page as tiles or as a list.

Click the appropriate button to change how assets are displayed:



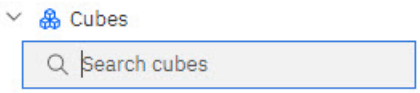

**1**
Tile view

**2**
List view

## Search at the node level in the data tree

To make it easier to locate an object within a long list of items in the data tree, you can now search at the node level in the tree.

When you expand the Cubes, Dimensions, Sets, Processes, Chores, or Control Objects:Cubes nodes in the data tree, you'll see a search box. For example, here's what the search box looks like under the Cubes node.

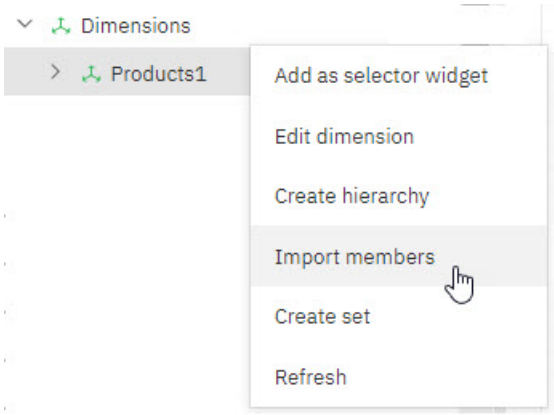



Start typing in the box to see the objects that contain your search string. The results are updated continuously as you enter characters.

## Import dimension members from the data tree

You can now right-click a dimension in the data tree and click **Import members** to initiate an import.

You no longer need to have the dimension open in a book to start importing members.




## Reset a sandbox

You can now reset data in a sandbox with a single click.

Click the **Sandbox discard** button to discard all data changes since your last commit action.




# 2.0.60 - What's new, January 12, 2021

Learn about new features and known issues in version 2.0.60 of IBM Planning Analytics Workspace in the following topics.

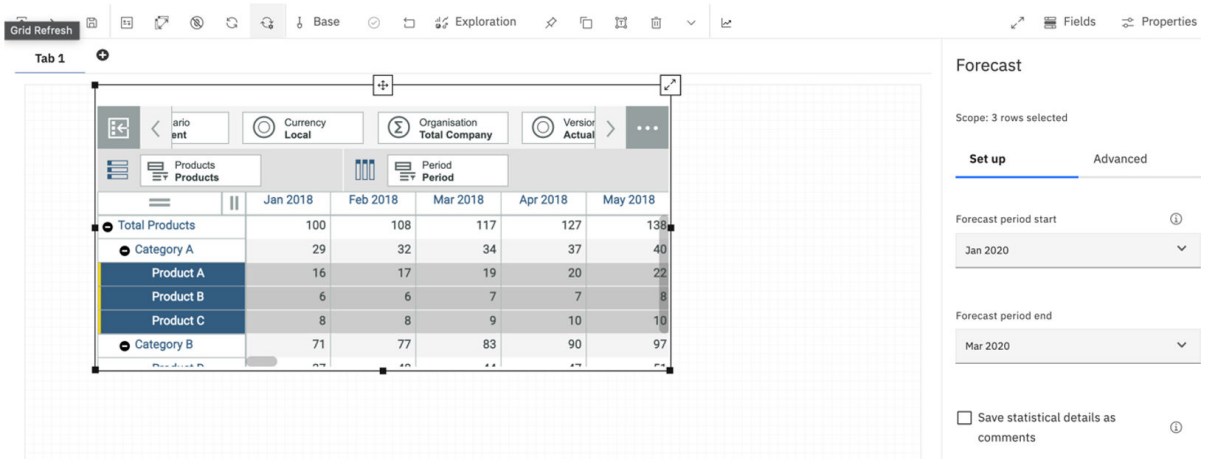
Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

## Forecasting: preview more than one row and forecast by using cube view data

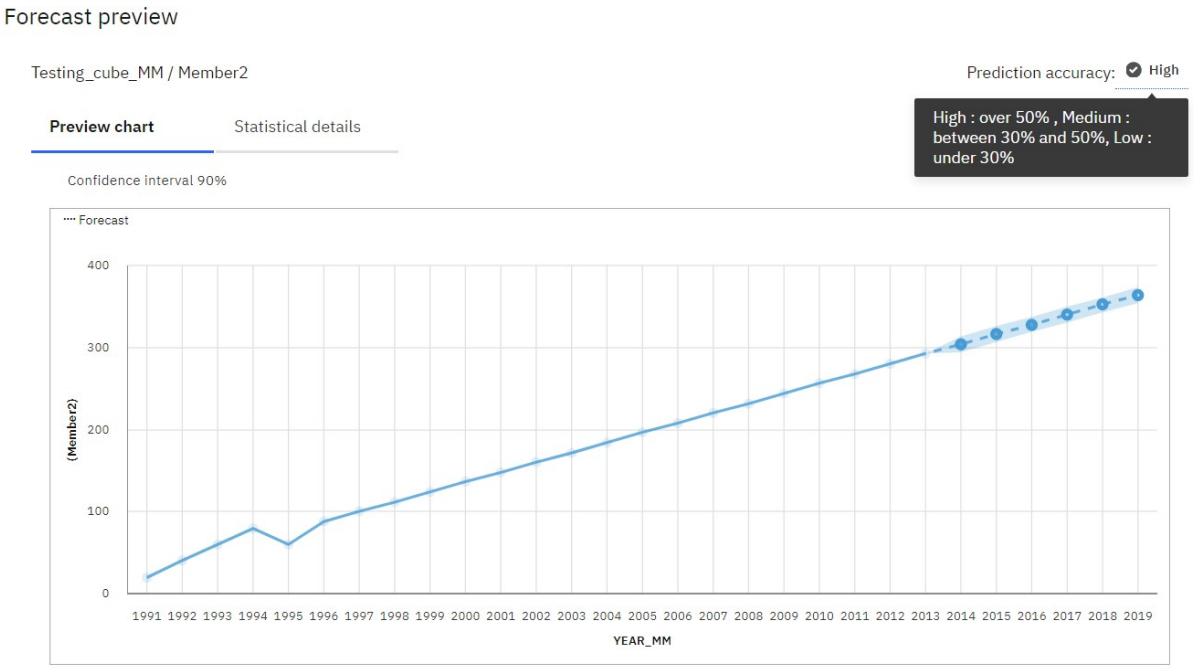
Forecasting in IBM Planning Analytics Workspace allows previewing based on up to 25 rows. And you can forecast by using cube view data.

### Preview more than one row

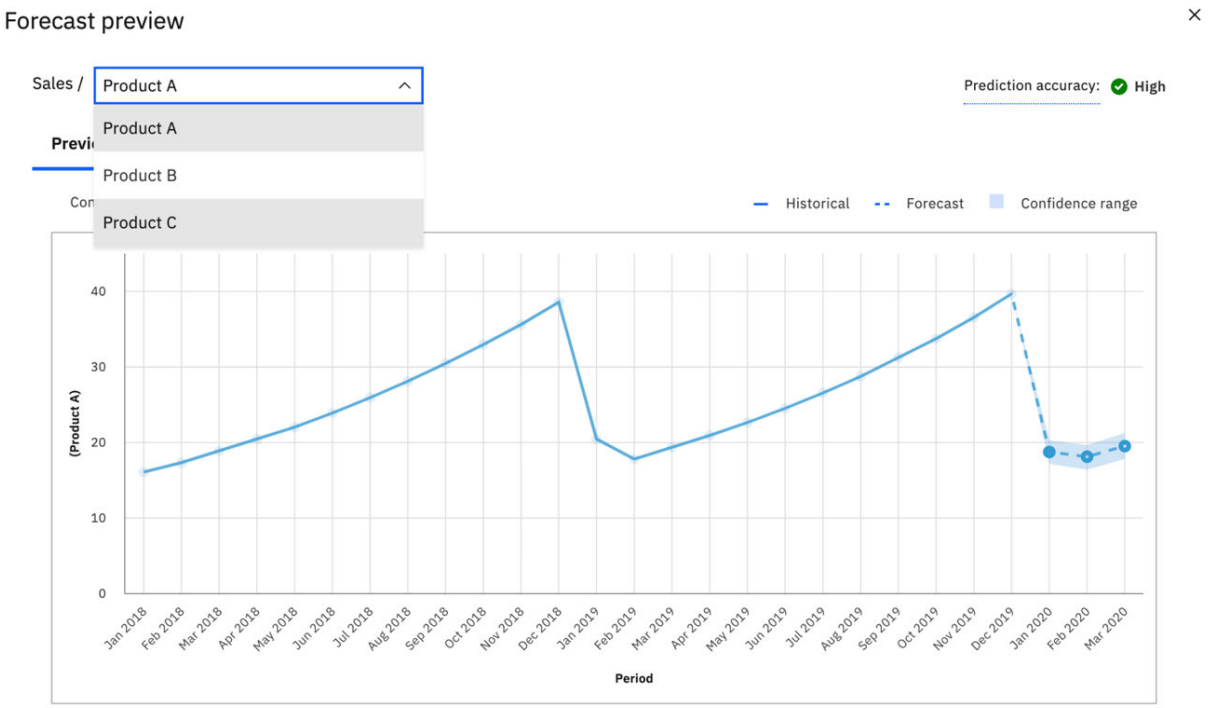
We now support the preview of up to 25 rows. In the following case, three rows are selected for preview and **Preview** is enabled.




Before SC60



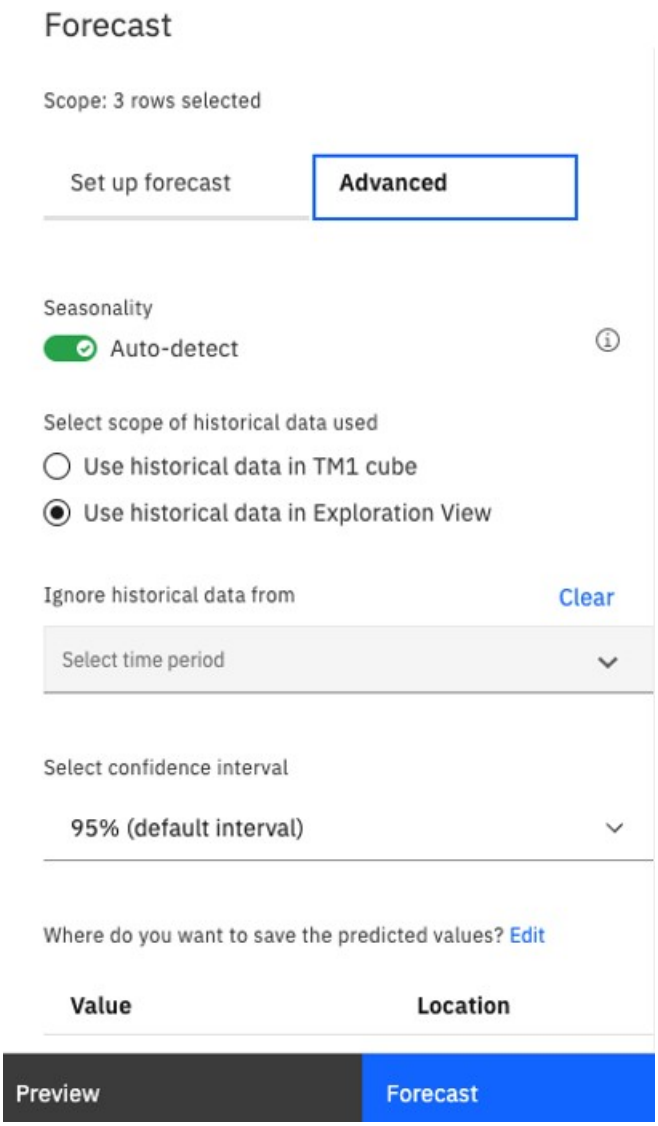

As of SC60




For more information, see Previewing more than one row.

**Forecast by using cube view data**

Initially, when the forecast preview is run, the data in TM1 is used as the historical data regardless of what is presented in the cube view. The reason for that was to include more history in the prediction, which results in higher accuracy. In SC60, a new option is added that allows you to choose whether to use the data in TM1 or the current cube view as the history.




In the **Advanced** tab in the **Forecast** window, you can select two options **Use historical data in TM1 cube** and **Use historical data in Exploration View**. The default is **Use historical data in Exploration View**.

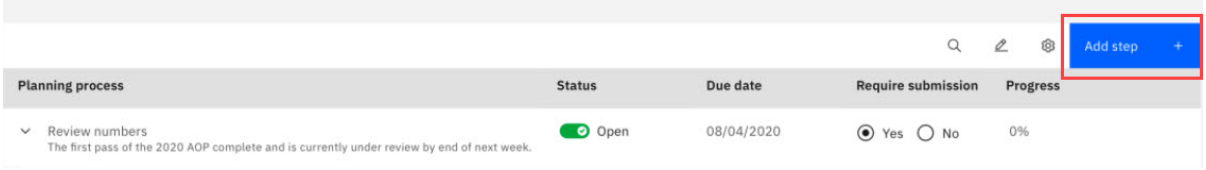
For more information, see Forecasting options.

For complete details on using forecasting in Planning Analytics Workspace, including a complete tutorial, see Forecasting.

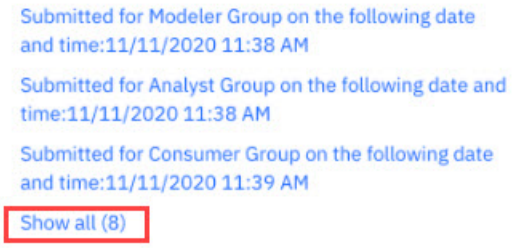
## Applications and Plans UI enhancements

The user interface for Applications and Plans has been updated to simplify administration.

The text link to Add a step (for plans) and Add a section (for applications) has been moved from the bottom of the plan/application grid and converted to a button at the top of the grid. This example shows the change in a plan, but you'll see the same change (with a different button label) for applications.



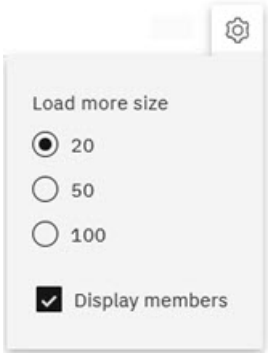
Additionally, the plan contribution panel has been modified to show only the first three submissions for a step. You can click **Show all** to reveal all the submissions.



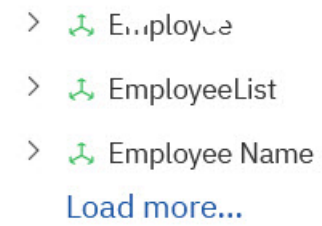

After reviewing the submissions, you can click **Show less** to collapse the list and show just the first three submissions.

## New data tree Settings menu

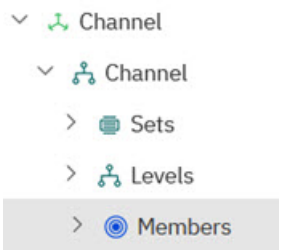
A new Settings menu lets you manage the behavior of the data tree.




You can specify a **Load more size** value to determine how many items of any object type are displayed in the data tree before you have to click **Load more** to retrieve and display more items. For example, here you can see a list of dimensions where the **Load more** option is enabled because the number or dimensions exceeds the specified **Load more size** value.




You can also use the **Display members** option to determine if the Members node appears in the data tree under dimension hierarchies. When the option is enabled, the Members node is available, as in this example.




When the option is disabled, the Members node is not available in the data tree.

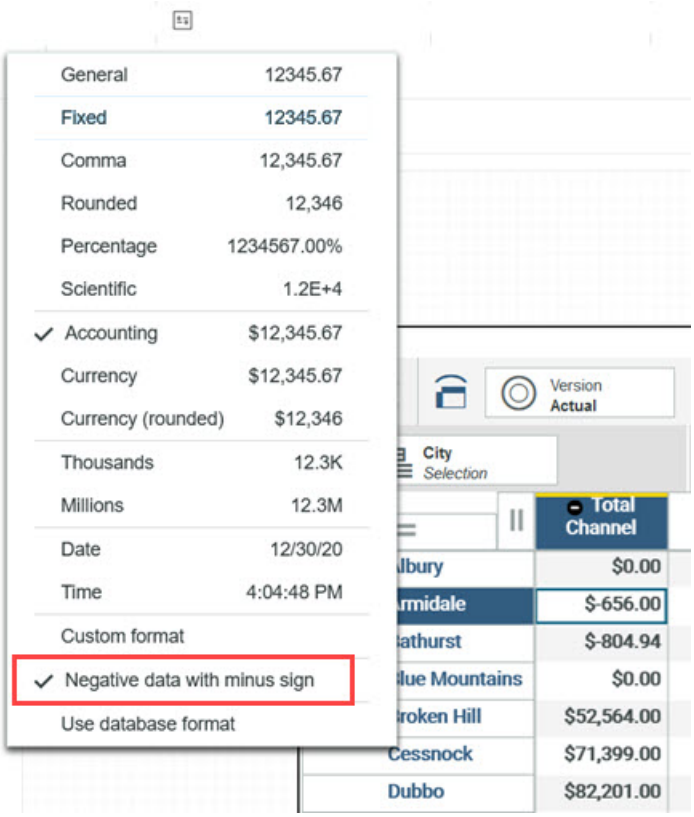
## Prefix negative values with a minus sign

When defining a display format for an entire view, a single row, a single column, or a cell widget, you can set the format to display negative values with a minus sign, rather than enclosed in parentheses.

The **Negative data with minus sign** option is available when you select any of the following formats:

- Comma
- Rounded
- Accounting
- Currency
- Currency (Rounded)
- Thousands
- Millions

This example shows the **Negative data with minus sign** option applied to the Accounting format and visible in a cube view.




## Delete a folder for a removed user

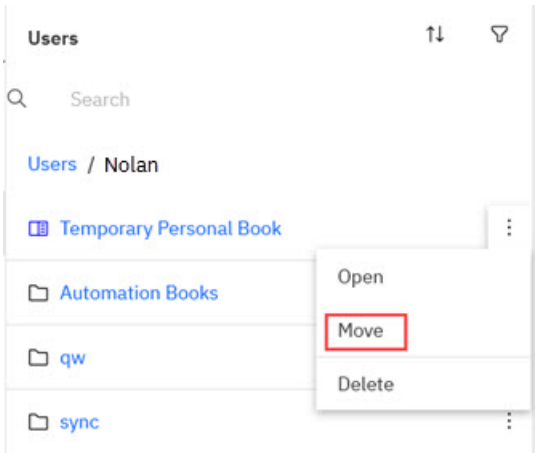
Previously, when a user was removed from Planning Analytics Workspace, the user's personal folder was renamed to Unknown and the folder could not be deleted by an administrator.

Administrators can now delete the Unknown folder associated with any deleted user.

## Move assets or folders in a user's personal folder

A **Move** option is now available for all assets and folders within a user's personal folder. A user or administrator can use this option to move an asset or folder to a new location, either in the Shared folder or a different place in the user's personal folder.

When you click the **Options** menu next to any asset or folder within a user's personal folder, you'll see the new **Move** option.




When you click the **Move** option, you can select a destination folder for the asset or folder. The destination can be in the Shared folder or in the user's personal folder. If you move to the Shared folder, you are prompted to set user/group permissions to the asset or folder.

# 2.0.59 - What's new, November 17, 2020

Learn about new features and known issues in version 2.0.59 of IBM Planning Analytics Workspace in the following topics.

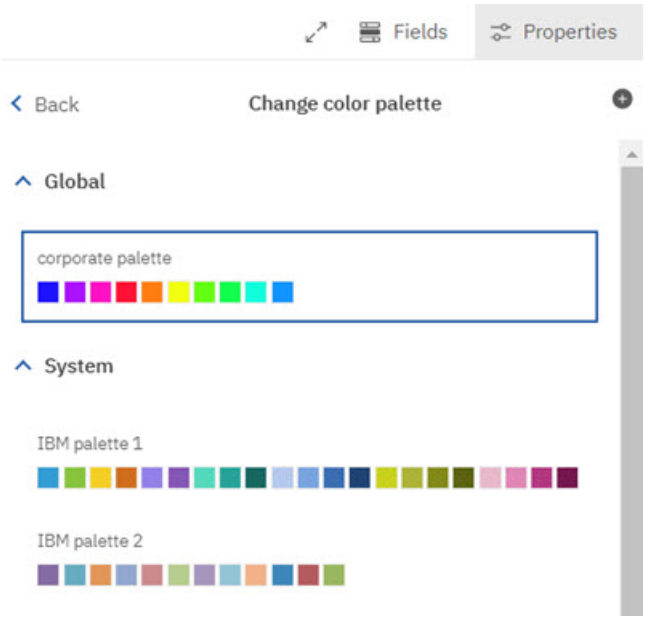
Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

## Create a global color palette

Administrators can create global color palettes that can be applied to books and visualizations in Planning Analytics Workspace.

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace Home page.
2. Click the **Excel and Customizations** tile.
3. Click the **Palettes** tab.
4. Click **Add global palette** +.

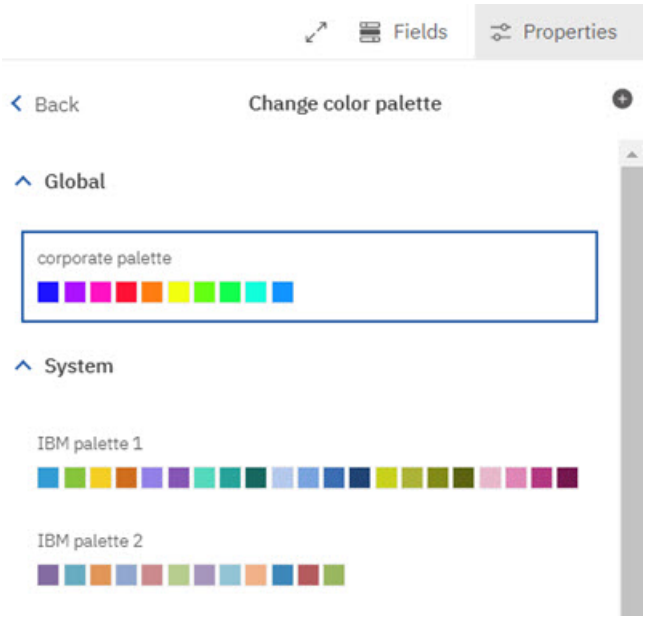



You can create the following types of color palettes:

**Categorical**
Used for visualizations that support discrete colors, like a bar or pie visualization.

**Continuous**
Used for visualizations that support color transitions, like a map or a heat map visualization.

After the global color palette is saved, users can apply the palette from the Global color palette properties.




For complete details on creating a global palette in Planning Analytics Workspace, see Create a global color palette.

## Support for iPad

The Planning Analytics Workspace new experience is now supported on iPad.

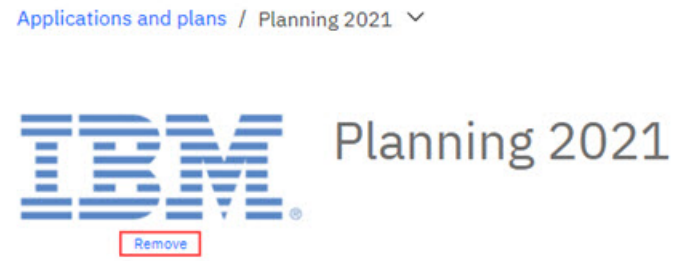
For details on mobile device capabilities and performance considerations, see Accessing Planning Analytics Workspace from Apple iPad.

# Usability enhancements

This release includes several usability enhancements.

## Remove a logo from an application or plan

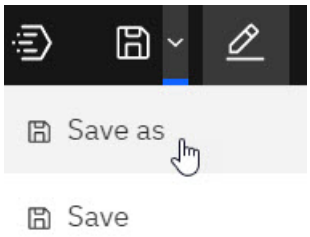
You can now delete a logo from an application or plan by clicking the **Remove** option directly beneath the logo.



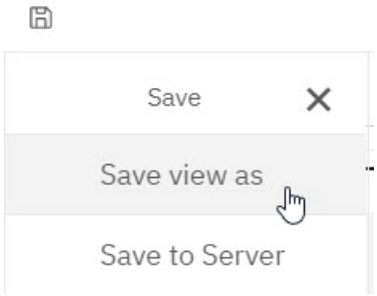

Delete option for logo

## 'Save as' defaults to current folder for books and views

If a book has previously been saved, then the folder where the book exists is the default folder when you choose **Save** > **Save as**.

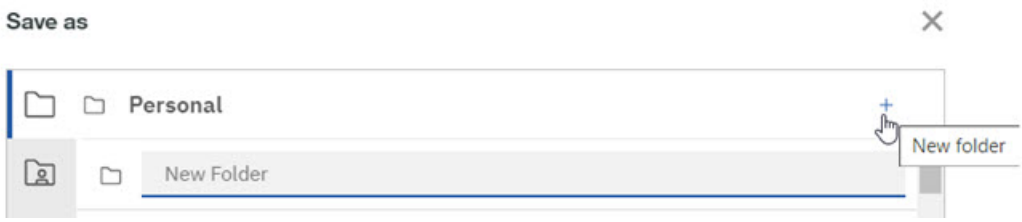



Similarly, if a view has previously been saved, then the folder where the view exists is the default folder when you choose **Save** > **Save view as**.




## Create a new folder when saving a book or view

When saving a book or view, you can now create a new folder to accept the book or view. You can create a new folder in the Shared folder, in your personal folder, or in any sub-folder of your personal folder. To create a new folder, click the New folder icon on the parent folder where you want to insert the new folder, then assign a name for the folder and click Save.



## Planning Analytics for Microsoft Excel: login or server access may fail in new experience

This issue may occur only when authenticating against a local TM1 database in the Planning Analytics new experience. You will not encounter this issue when authenticating against a local database or any Planning Analytics on Cloud database.

When attempting to log in to a TM1 database from Planning Analytics for Microsoft Excel, you may receive the following error under certain configurations or conditions: `Cannot load the requested view`.

To work around this issue, you can dismiss the error and again select the desired TM1 database and continue to log in with a valid user name.

This issue will be corrected in an upcoming release.

# 2.0.58 - What's new, October 21, 2020

Planning Analytics Workspace 2.0.58 SC is a cloud release that includes significant changes in the look and feel of Planning Analytics Workspace, as well as important new functionality.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

## Learn from the experts

There are a lot of big changes in the Planning Analytics Workspace new experience, introduced in 2.0.57 local and 2.0.58 cloud. From a completely revised interface to significant new features like forecasting, the best place to learn about these changes is from the detailed blogs created by our subject matter experts.

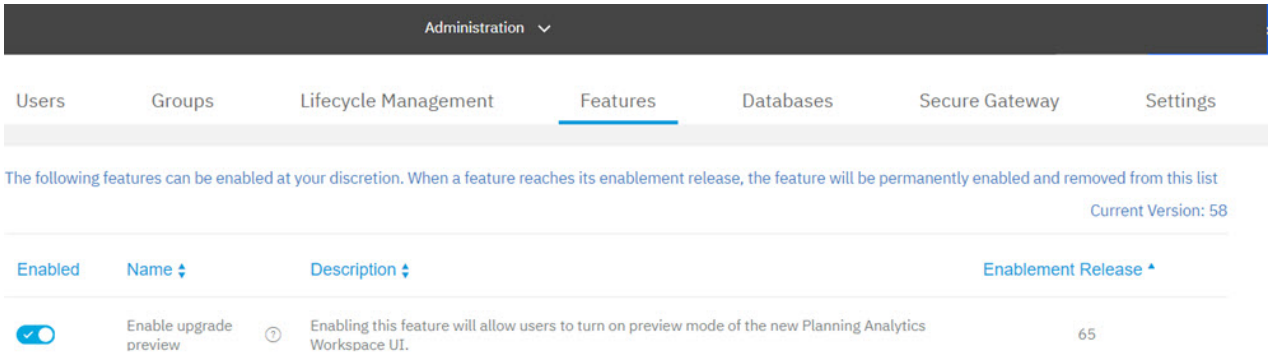
Click here to visit the central blog about Planning Analytics Workspace enhancements and new capabilities. From this central blog, you'll find links to other blog posts and additional information from a range of Planning Analytics Workspace pros.

## Manage the new experience for your users

To make the move to the new interface more manageable, Planning Analytics Workspace administrators can manage when a new experience preview is exposed to their users and when a permanent upgrade to the new experience is executed.

### Enable the preview

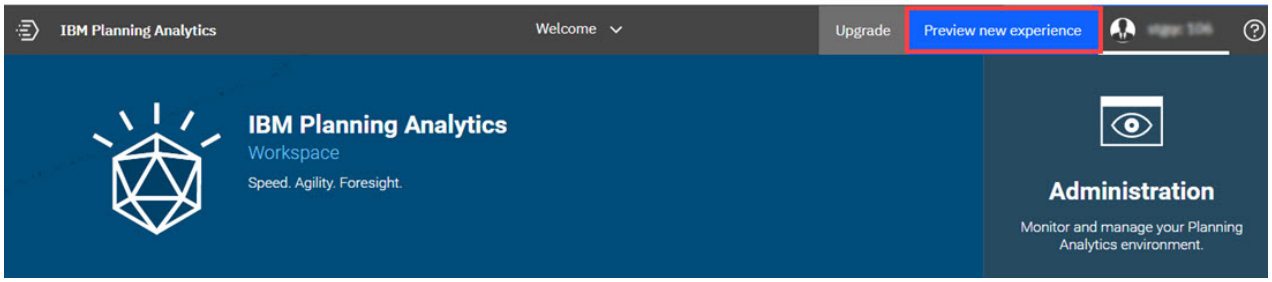
To make a new experience preview available for your users, enable the **Enable upgrade preview** feature on the Features administration tab.



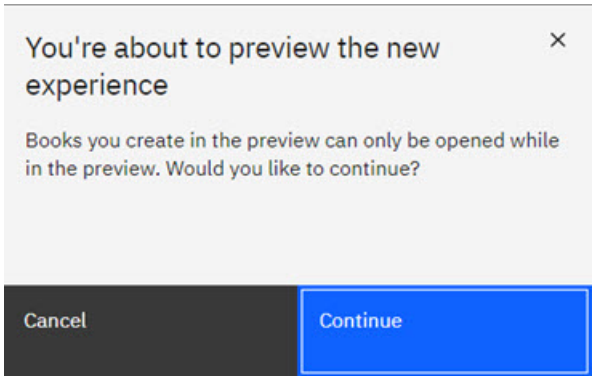

As with all features that are subject to administrator enablement, the feature is enabled by default. If you don't want the new experience preview to be available to your users, disable the feature. Note that disabling the feature also removes the Upgrade option for administrators.

See Manage features on IBM Knowledge Center for full details on using the Features administration tab.

When the feature is enabled, all users see a **Preview new experience** button on the Planning Analytics Workspace banner. (Note that the adjacent **Upgrade** button is visible only to administrators. We'll discuss that further in a moment.)




You can click the button to experiment with the new experience. When you click the button, you'll be asked for confirmation. Click **Continue** to proceed to the new experience preview.




While using the new experience preview, all users should be aware of these considerations:

- You can return to the classic Planning Analytics Workspace interface at any time by clicking the **Return to Classic Experience** button on the banner.
- Books created in the classic experience can be opened in the new experience. When you open a book that was created in Planning Analytics Workspace Classic, a message appears saying: "Your Dashboard has been upgraded. Save now to maintain optimum performance."
- When you open a book that was created in Planning Analytics Workspace Classic, you should use Save As and save the book to a new name or location in order to create a copy of the classic book and take advantage of the features in the new experience. In this scenario, the original book is still visible in the classic experience and the copy is available in the new experience, making it easy to compare and validate your assets.
- Any books saved in the new experience cannot be viewed in the classic experience. If you attempt to open such a book, you'll receive an error message. Views saved to the content store in the new experience cannot be opened in Planning Analytics for Microsoft Excel.

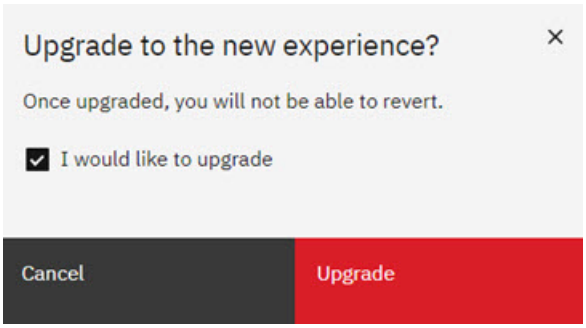
## Perform a permanent upgrade to the new experience

Once all your users are comfortable with the new interface, have validated their current assets, and are ready to adopt the change, an Administrator can perform a permanent upgrade to the new experience.

**Note:** Remember there is no way to undo an upgrade, so please be sure to complete all of your testing before proceeding with a permanent upgrade to the new Planning Analytics Workspace experience. To ensure consistency and a smooth upgrade, we recommend that you preview and upgrade first in a non-production environment, then in a test environment (if applicable), and finally in your production environment.

To perform the upgrade, click the **Upgrade** button on the Planning Analytics Workspace Classic banner.

To avoid an accidental upgrade, you'll be prompted for confirmation. Select the **I would like to upgrade** option, then click **Upgrade**.




After the upgrade is complete, the **Preview** and **Upgrade** buttons will no longer be visible and all future logins take all users directly to the new experience.

After upgrading, users must open and save any existing books before they can take advantage of the enhancements available in the new experience.

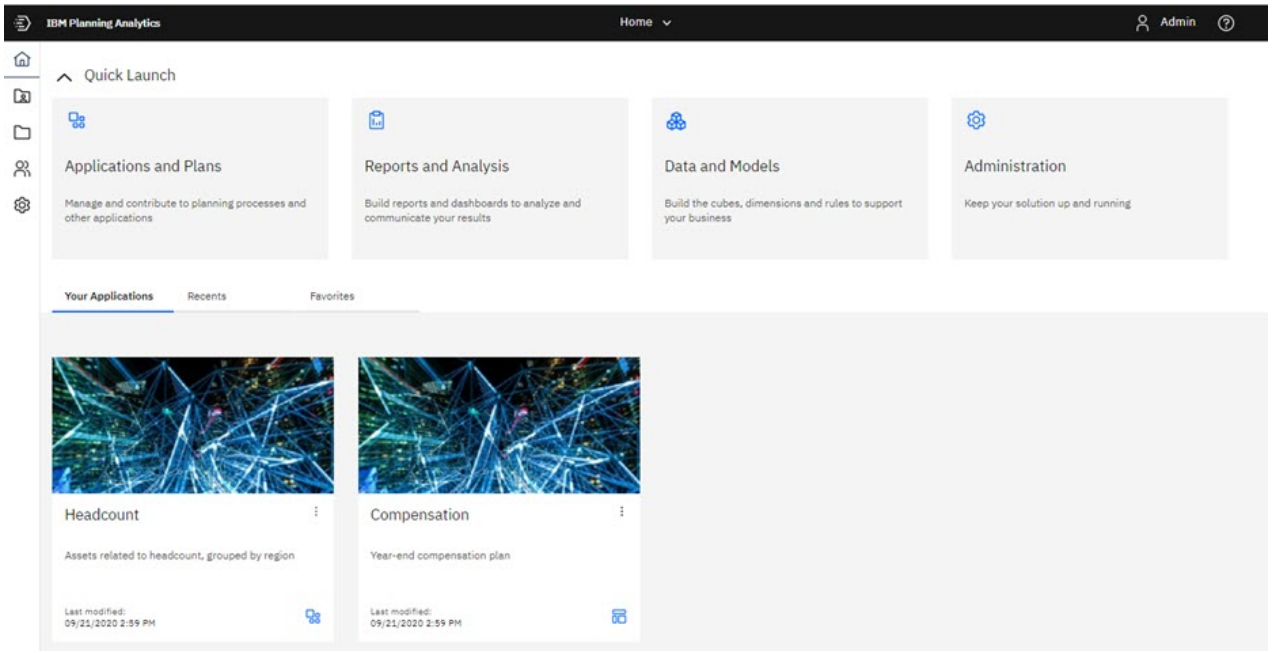
## Improved look and feel

The Planning Analytics Workspace user interface has been updated to make it easier to accomplish tasks, provide a more consistent experience with other IBM products, and to simplify the transition between Planning Analytics Workspace and Cognos Analytics.

As part of the interface update, new icons adhering to the Carbon Design principles have been introduced throughout Planning Analytics Workspace. For more information about Carbon Design principles, see https://www.carbondesignsystem.com.

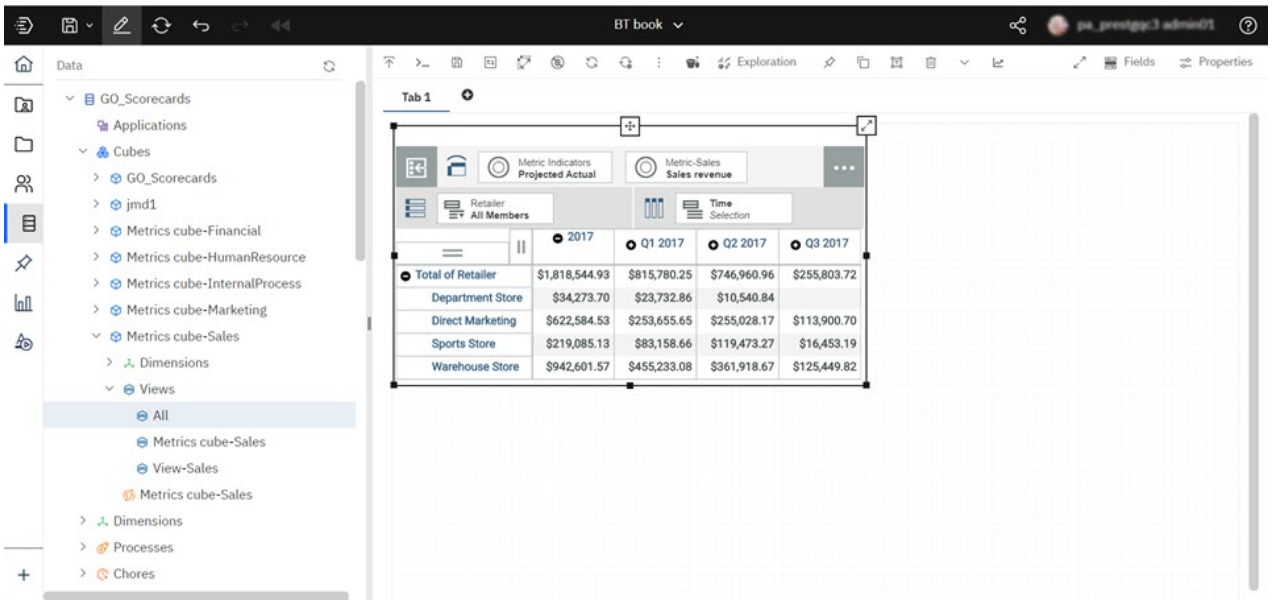
**Note:** Because the interface that exists in versions of Planning Analytics Workspace prior to 2.0.57 SC is still supported, it is necessary to differentiate between the new interface and the 'old' interface in the documentation. When there is a divergence of procedures or capabilities between the current (new) interface and the old, the documentation describes the current interface as Planning Analytics Workspace and describes the interface in 2.0.55 SC and prior versions as Planning Analytics Workspace Classic.

A new **Home** page for Planning Analytics Workspace is the first significant changes you'll notice. On the new **Home** page, you can quickly access the area you want to work in, customized for your role within Planning Analytics Workspace. You can also quickly open your applications and plans, as well as your recent and favorite items.




The prominent Quick Launch tiles that provide immediate access to **Applications and Plans**, **Reports and Analysis**, **Data and Models**, and **Administration** are dynamically displayed depending upon your role when you log in to Planning Analytics Workspace. Only the tiles that you can use based on your role appear. For example, an administrator sees all of the Quick Launch tiles, while an analyst sees only **Applications and Plans** and **Reports and Analysis**. You can click the Quick Launch show/hide button Quick Launch to show or hide the Quick Launch tiles. When you hide the tiles, you can see more of your applications, recent items, or favorite items.

You'll also notice improvements in other familiar places. For example, in books the user interface is simplified, new icons are present, and the toolbar has been decoupled from the view.

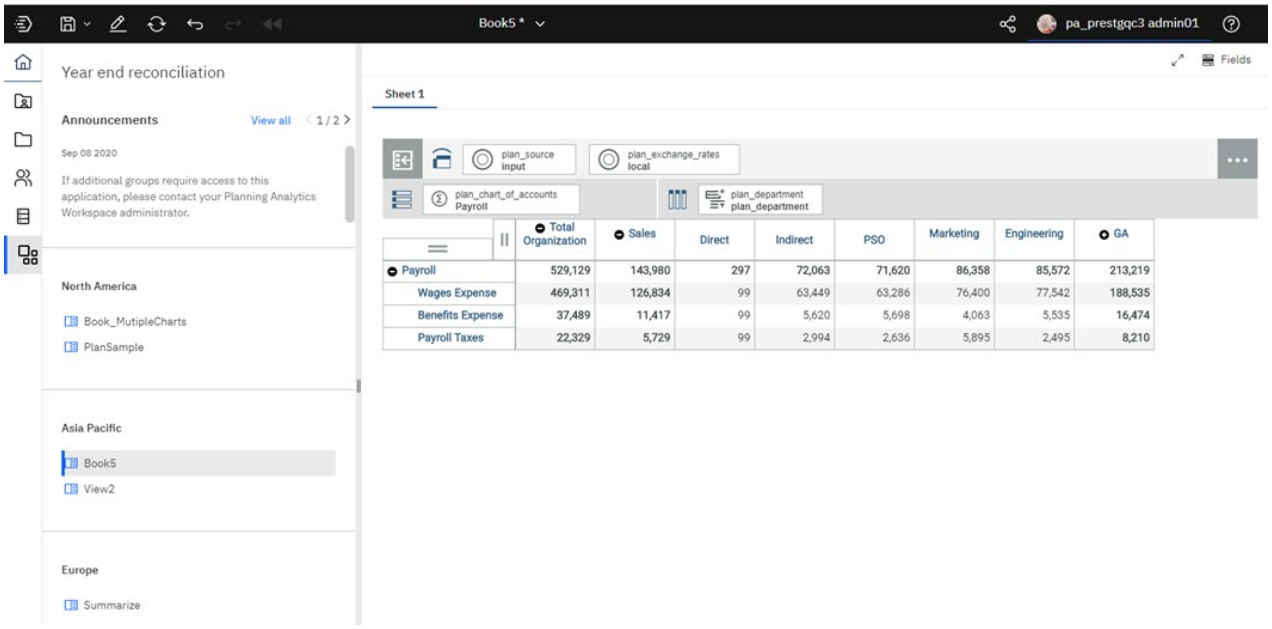



Use of the new interface is described in the relevant topics throughout the Planning Analytics Workspace help and documentation.

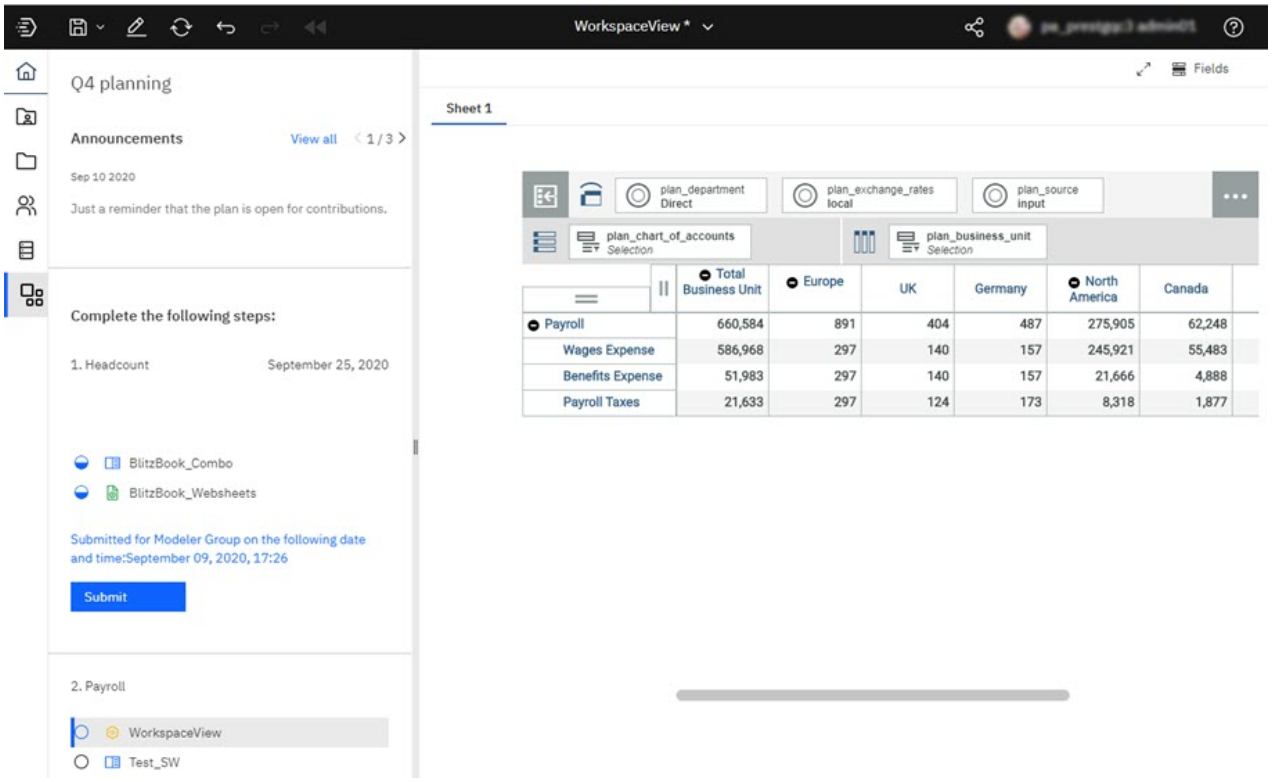
# Applications and Plans

Applications and plans let you organize logically related Planning Analytics Workspace assets such as books, view, and websheets in containers.

An application contains related assets that are grouped in sections. These sections might reflect the structure of your organization, planning and budgeting requirements, or any other relevant grouping of assets. While an application contains logically related assets, there are no implied or required actions associated with the assets or sections in an application. An individual asset can belong to more than one application.




A plan contains assets that are grouped in steps. These steps can represent discrete tasks or contributions that must be completed in a planning or budgeting process. While steps can be ordered in a plan, there is no requirement for contributors to complete the steps sequentially; they can be completed in any order. Steps can also be assigned a due date for contributions. An administrator can require that steps be explicitly submitted for approval, and an administrator can reject and reset a submission.

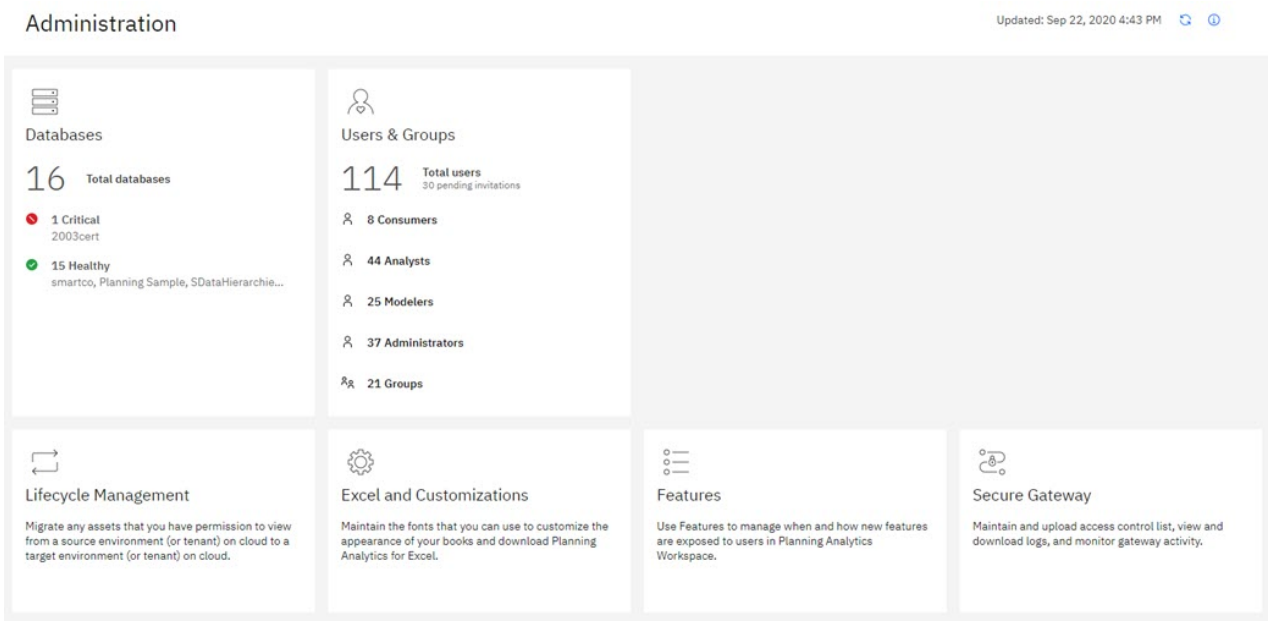



For complete details on creating, managing, and using applications and plans, see Applications and plans.

## Administration page changes

The Planning Analytics Workspace **Administration** page has been reorganized to provide greater insight into your environment and to simplify access to administrative tasks.

The Administration page now includes several task-specific tiles. The Databases tile provide quick insight into the health of your databases. The Users and Groups tile lets you know how many users are assigned to each role and the number of groups that are defined.




You can click any tile to perform the administrative tasks associated with the tile.

The tiles available on the **Administration** page vary depending on whether you are running Planning Analytics Workspace Local or on Cloud. The Agents tile is available only on Local, while the Secure Gateway tile is available only on Cloud.

For complete details on using the new Administration page, see Administer Planning Analytics Workspace.

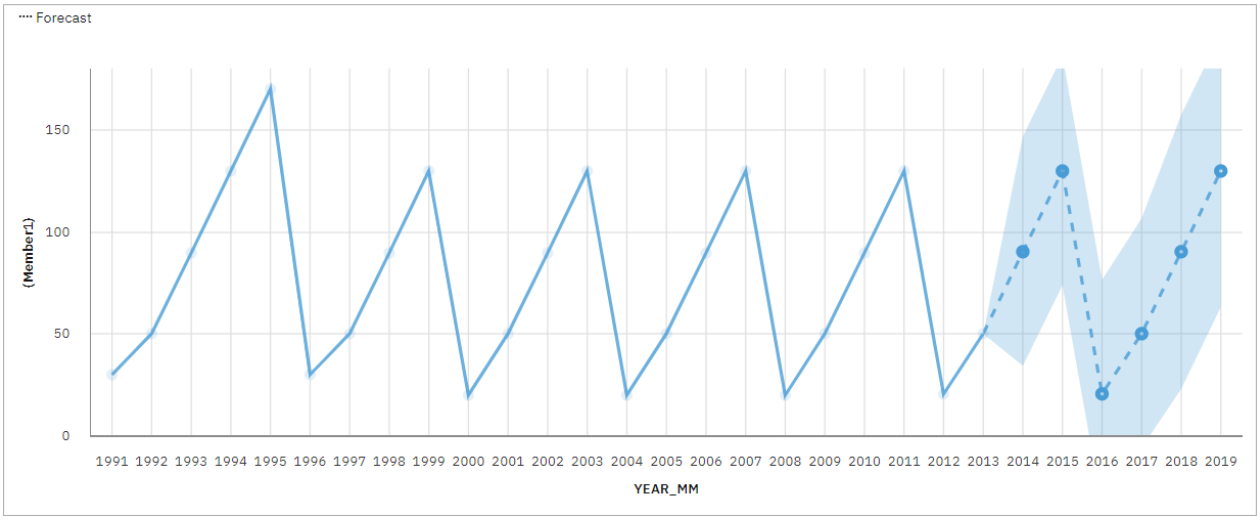
## Forecasting

You can now use forecasting in IBM Planning Analytics Workspace to discover and model trend, seasonality, and time dependence in data.

You can forecast in Planning Analytics Workspace by using automated tools that model time-dependent data. Automated model selection and tuning makes forecasting easy to use, even if you are not familiar with time series modeling.

Forecasts and their confidence bounds are displayed in visualizations as a continuation of historic data. You can also view the statistical details for generated models if you want to see the technical background.

The following example shows forecasting values and confidence bounds in a line visualization.




Specifying time series in forecasts often requires data manipulation. Planning Analytics Workspace supports a wide range of time series without the need for manipulation, ranging from standard date and time types, to nested periodic and cyclical time fields. When data is recognized as a time series, data preparation is automated. Appropriate trend and seasonal periods are detected, and models are selected from a set of nine different model types.

You can forecast in line visualizations. Forecasting allows analysis of hundreds of time series per visualization. Forecasts and confidence bounds are computed for each time series, and displayed in the visualization as extensions of the current data. You can inspect each time series separately, and tailor the forecast and results to your own data and requirements.

If you are familiar with forecasting models, you can view the selected model type, estimated model parameters, standard accuracy measures, and processing summary information.

For complete details on using forecasting in Planning Analytics Workspace, including a complete tutorial, see Forecasting.

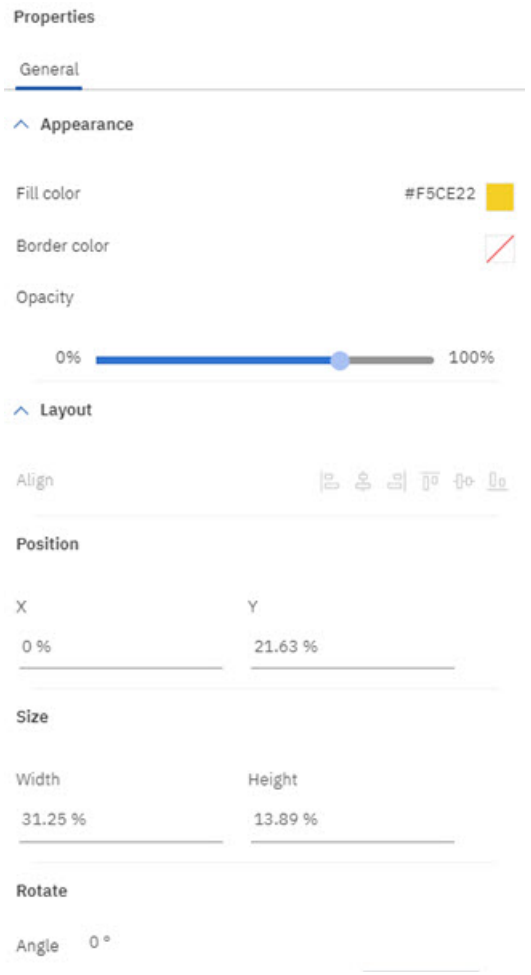
# Book and visualization improvements

The properties available to manage all aspects of your books have been significantly expanded. The visualization types available to use in Planning Analytics Workspace have been updated to provide more and improved visualizations.

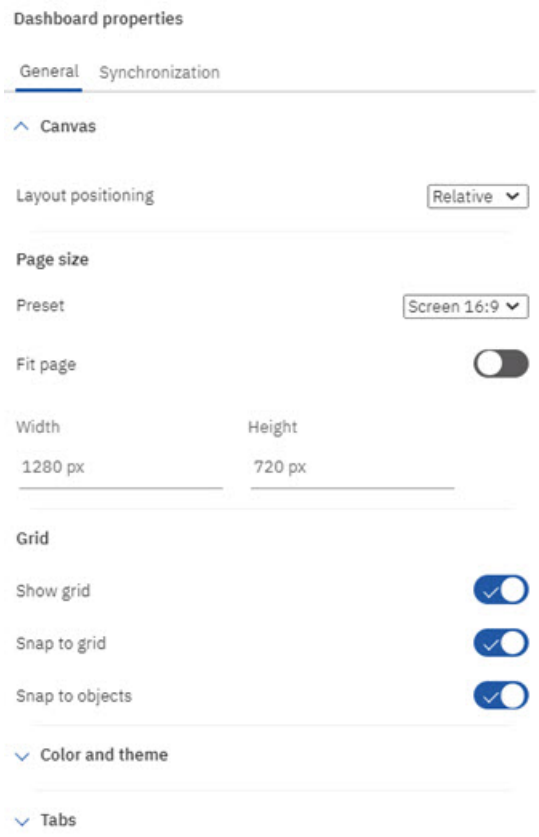
## Books

All objects in a book, such as explorations, visualizations, buttons, images, and text, can now be precisely managed using an expanded selection of properties.

When you select an object in a book and then click the **Properties** tab, you'll see an expanded list of properties that you can set to manage the object in your book. The properties available vary by object type, and the properties you're familiar with are still available, but new properties allow you to precisely manage the size, position, alignment, and appearance of all objects in a book.



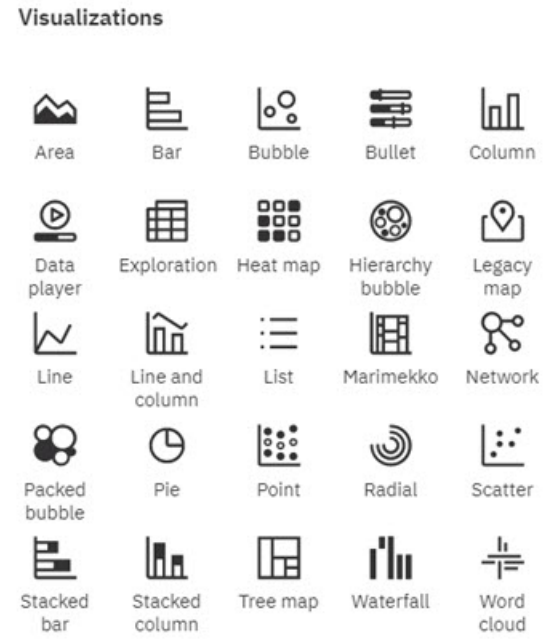

You can also set Dashboard properties to manage the general appearance of your book.



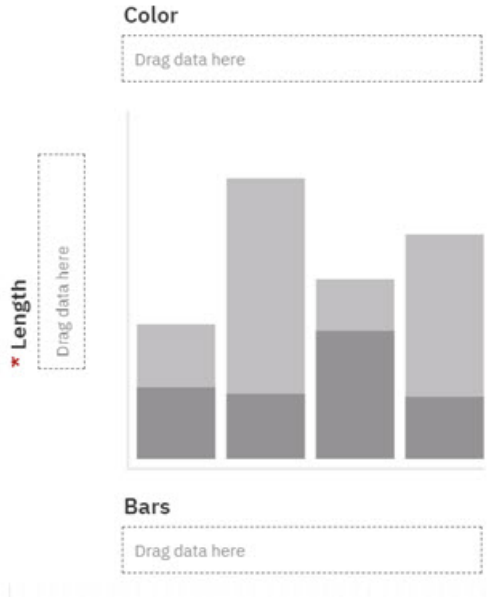

Details on using these properties are provided in the individual topics that describe how to build and manage books. For more information, see Work in books and views.

## Visualizations

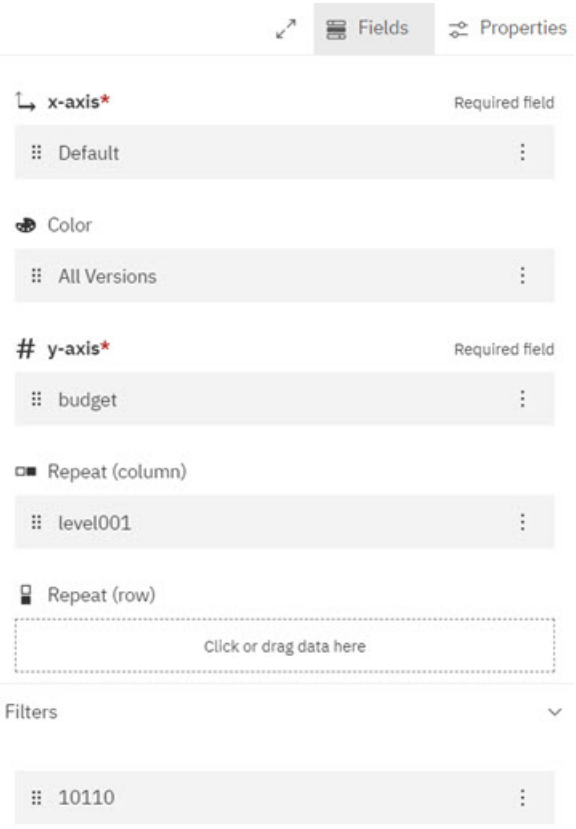
The list of visualizations available in Planning Analytics Workspace has been expanded to include twenty five options.



Explorations can be changed to visualizations in the same way you've always done it, but you can also place an empty visualization onto a book and use drop zones to build the visualization.




You can drag and drop dimensions from the Data tree onto the **Drag data here** drop zones to build a visualization from scratch. Once the visualization is complete, you can manage the visualization using the Fields tab on a book. You can drag and drop dimensions to different fields to change the structure of your visualization or you can click a field to select a new member to use in your visualization.

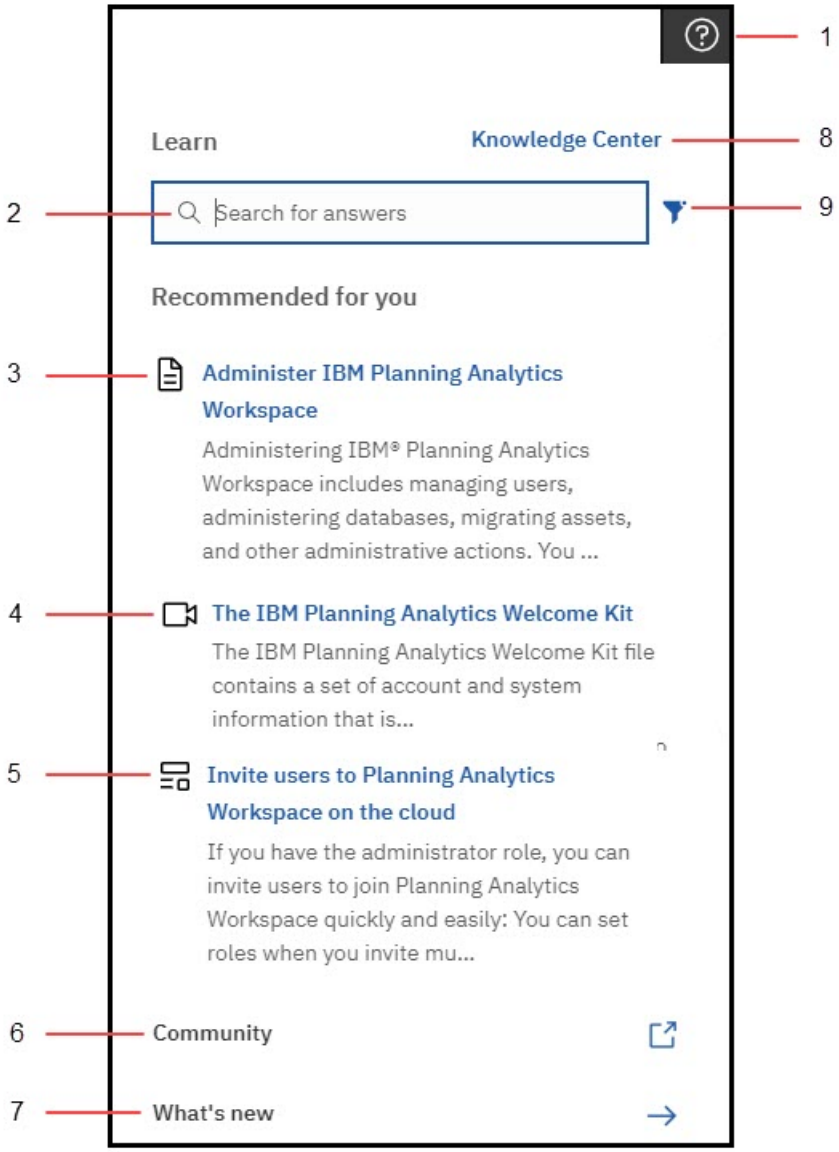



For further details, see Visualizations.

# Ask for help in the cognitive Learn pane

If you want to learn more about how to use Planning Analytics Workspace, click the Help icon and ask a question. The cognitive help tailors your help experience based on where you are working in the product and finds only the answers that are relevant to your user role. You can find the latest videos, blogs, and documentation.

Use these features on the Learn pane to find answers and be more productive!




**1**

Click the Help icon to open the Learn pane (it remembers where you were the last time you opened it). Click anywhere to close it. The Learn pane recommends content that relates to your task and finds similar content that you might also like. And, it is always learning! When you search and find answers, you are training the Learn pane and those answers influence future recommendations.

**2**

Type a question in the Search box. You can search in any supported language in the Learn pane and you see translated documentation in your search results. You also see blogs and videos that match your search, however, blogs and videos aren't translated.

**3**

Read the formal product documentation, sourced from IBM Knowledge Center.

4. Watch a video! Sometimes the best way to learn is to see it in action.
5. Read a post in the Planning Analytics Community Blog The community blog posts are written by experts who use Planning Analytics Workspace and share their tips and tricks.
6. Go to the Planning Analytics Community. In the community, you can find the latest articles, blog posts, and events. You can also start and contribute to discussions about Planning Analytics.
7. Click **What's New** to find out what is new in the latest release of Planning Analytics Workspace.
8. Visit the IBM Knowledge Center for all IBM Planning Analytics documentation In the IBM Knowledge Center, you can read all documentation, including related products.
9. Filter your search results to show only your preferred content type: videos, blogs, or documentation.

# Considerations for upgrading to Planning Analytics Workspace new experience

Users should be aware of the following considerations before upgrading to the new experience in Planning Analytics Workspace 2.0.57 SC local and 2.0.58 cloud.

The upgrade to Planning Analytics Workspace 2.0.57 SC local is a permanent upgrade. There is no way to revert to a prior version of Planning Analytics Workspace Classic.

likewise, when an administrator commits to an upgrade in 2.0.58 cloud, there is no way to revert to the 'classic' user experience.

## Features not supported in initial 2.0.57 local/2.0.58 cloud releases

These features are not supported in the initial new experience Planning Analytics Workspace release. Support for these features may be reintroduced in subsequent releases.

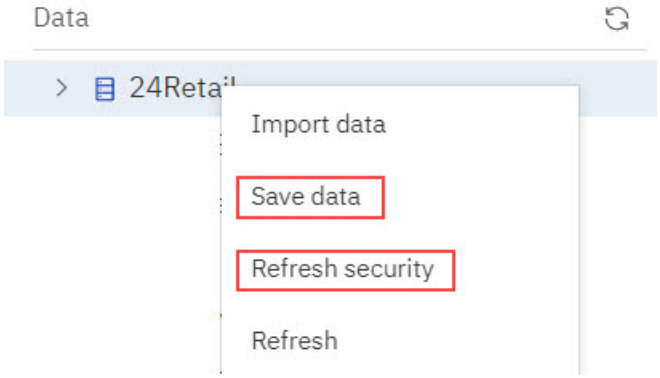
- Tree map visualizations upgrade successfully *except* when more than one dimension is present on an axis, in which case the tree map is converted to an Exploration during book upgrade.
- The Reports and Analysis landing page does not display individual tiles for books, views, or websheets. Rather, the Reports and Analysis landing page displays a searchable and sortable list of all assets.
- Mobile devices are not fully supported in the initial new experience release, as some gestures are not yet implemented.
- An Administrator cannot set up a global color palette. (This capability is supported as of Planning Analytics Workspace 2.0.59.)
- When you click **Share** > **Export**, you cannot share a book or view as an image or PowerPoint document. You can, however, export as PDF with enhanced print options.
- The Intent bar (sometimes called the NLP bar) is not available in this release.

## Differences in behavior between Planning Analytics Workspace 2.0.57 local/2.0.58 cloud and Planning Analytics Workspace Classic

- Chats are deprecated, as previously announced in this deprecation notice.
- Bookmarks and history are no longer available on the Data tree. Instead, you can use the Recents or Favorites tabs on the Planning Analytics Workspace Home page to open assets that you have recently viewed or favorited.
- Collections has been changed to Pins. You can pin a view or websheet from a book. You can access pinned items from the Pin button while in Edit mode.

## Database administration enhancements

You can now save data and refresh security for a selected database directly from the Data tree.




### Save data

To save all data for a database from memory to disc, right-click the database in the Data tree, then click **Save data**. This action also restarts the `tm1s.log` file for the database.

### Refresh security

To update all security structures and assignments for a database, right-click the database in the Data tree, then click **Refresh security**.

## Planning Analytics for Microsoft Excel: login or server access may fail in new experience

This issue may occur only when authenticating against a local TM1 database in the Planning Analytics new experience. You will not encounter this issue when authenticating against a local database or any Planning Analytics on Cloud database.

When attempting to log in to a TM1 database from Planning Analytics for Microsoft Excel, you may receive the following error under certain configurations or conditions: `Cannot load the requested view`.

To work around this issue, you can dismiss the error and again select the desired TM1 database and continue to log in with a valid user name.

This issue will be corrected in an upcoming release.

# 2.0.57 - What's new, October 2, 2020

Planning Analytics Workspace 2.0.57 SC is a local-only release that includes significant changes in the look and feel of Planning Analytics Workspace, as well as important new functionality.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in previous releases, see Chapter 1, “What's new in Planning Analytics Workspace,” on page 1.

## Learn from the experts

There are a lot of big changes in the Planning Analytics Workspace new experience, introduced in 2.0.57 local and 2.0.58 cloud. From a completely revised interface to significant new features like forecasting, the best place to learn about these changes is from the detailed blogs created by our subject matter experts.

Click here to visit the central blog about Planning Analytics Workspace enhancements and new capabilities. From this central blog, you'll find links to other blog posts and additional information from a range of Planning Analytics Workspace pros.

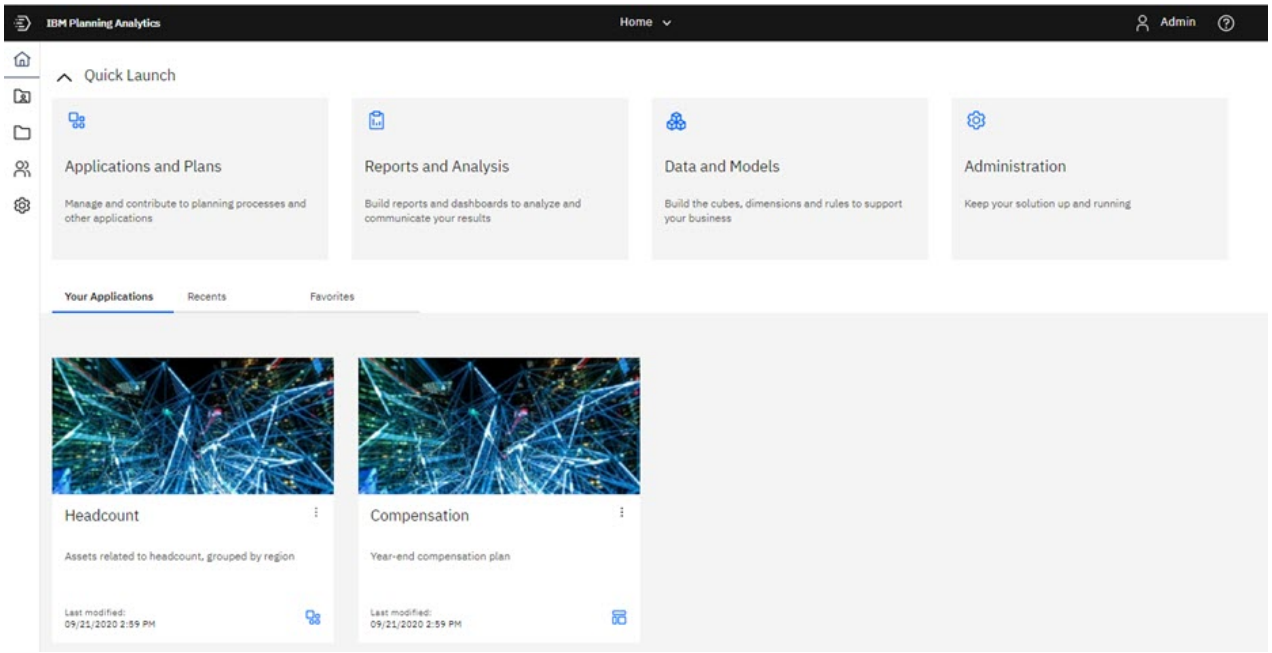
# Improved look and feel

The Planning Analytics Workspace user interface has been updated to make it easier to accomplish tasks, provide a more consistent experience with other IBM products, and to simplify the transition between Planning Analytics Workspace and Cognos Analytics.

As part of the interface update, new icons adhering to the Carbon Design principles have been introduced throughout Planning Analytics Workspace. For more information about Carbon Design principles, see https://www.carbondesignsystem.com.

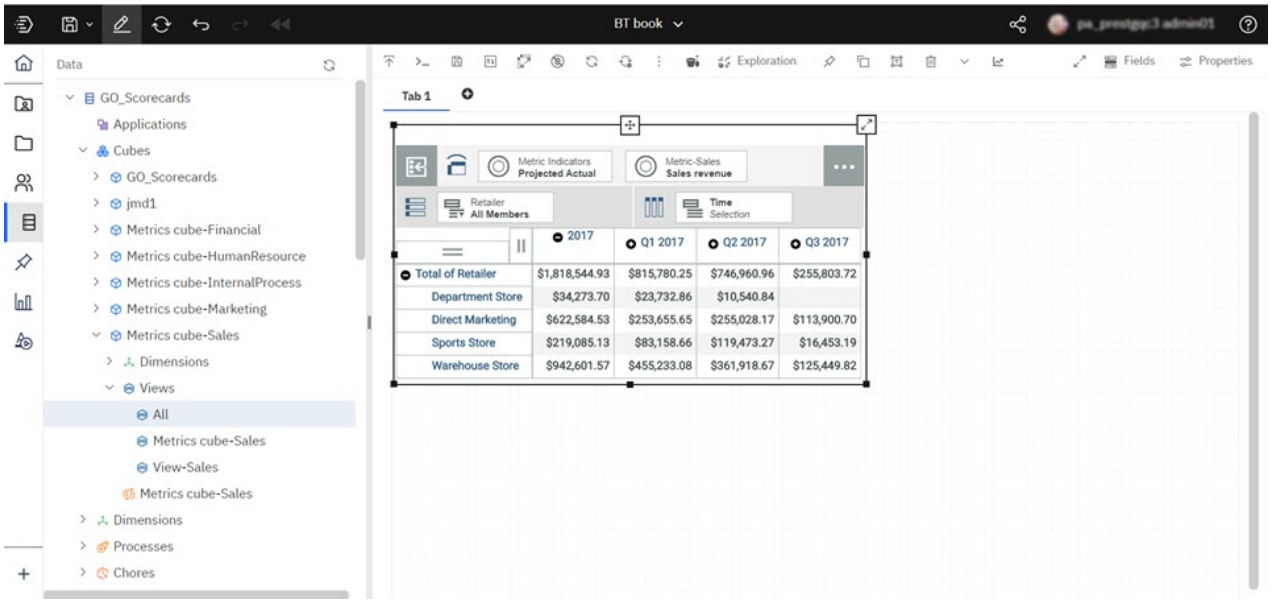
**Note:** Because the interface that exists in versions of Planning Analytics Workspace prior to 2.0.57 SC is still supported, it is necessary to differentiate between the new interface and the 'old' interface in the documentation. When there is a divergence of procedures or capabilities between the current (new) interface and the old, the documentation describes the current interface as Planning Analytics Workspace and describes the interface in 2.0.55 SC and prior versions as Planning Analytics Workspace Classic.

A new **Home** page for Planning Analytics Workspace is the first significant changes you'll notice. On the new **Home** page, you can quickly access the area you want to work in, customized for your role within Planning Analytics Workspace. You can also quickly open your applications and plans, as well as your recent and favorite items.




The prominent Quick Launch tiles that provide immediate access to **Applications and Plans**, **Reports and Analysis**, **Data and Models**, and **Administration** are dynamically displayed depending upon your role when you log in to Planning Analytics Workspace. Only the tiles that you can use based on your role appear. For example, an administrator sees all of the Quick Launch tiles, while an analyst sees only **Applications and Plans** and **Reports and Analysis**. You can click the Quick Launch show/hide button Quick Launch to show or hide the Quick Launch tiles. When you hide the tiles, you can see more of your applications, recent items, or favorite items.

You'll also notice improvements in other familiar places. For example, in books the user interface is simplified, new icons are present, and the toolbar has been decoupled from the view.

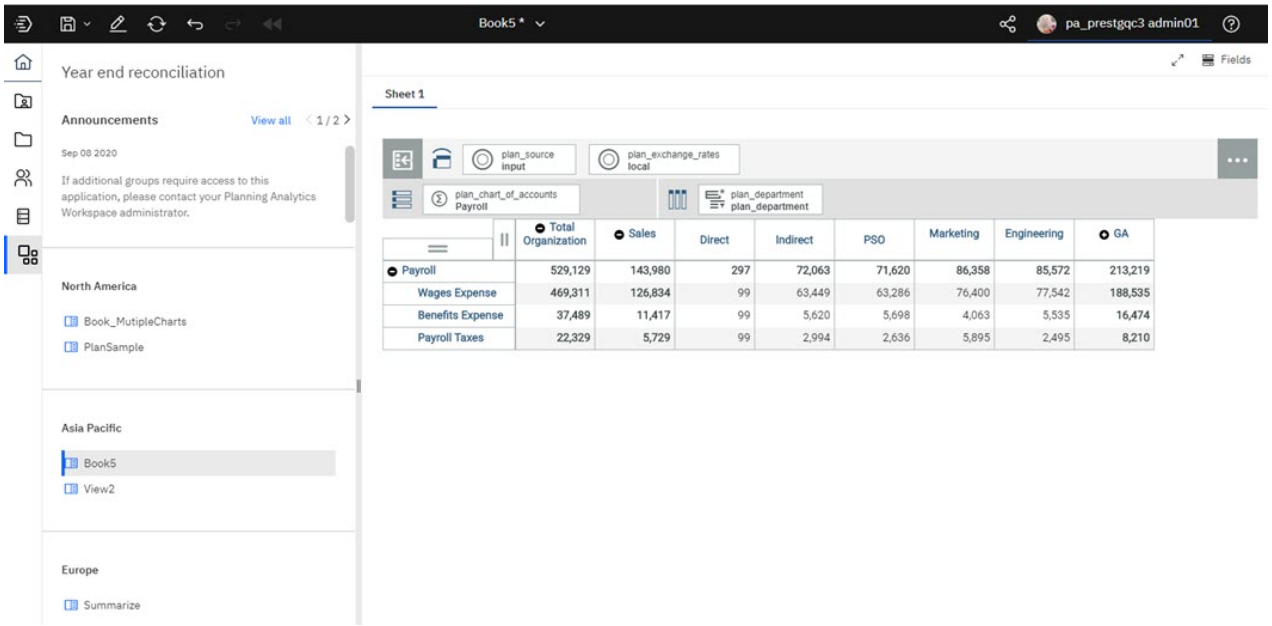



Use of the new interface is described in the relevant topics throughout the Planning Analytics Workspace help and documentation.

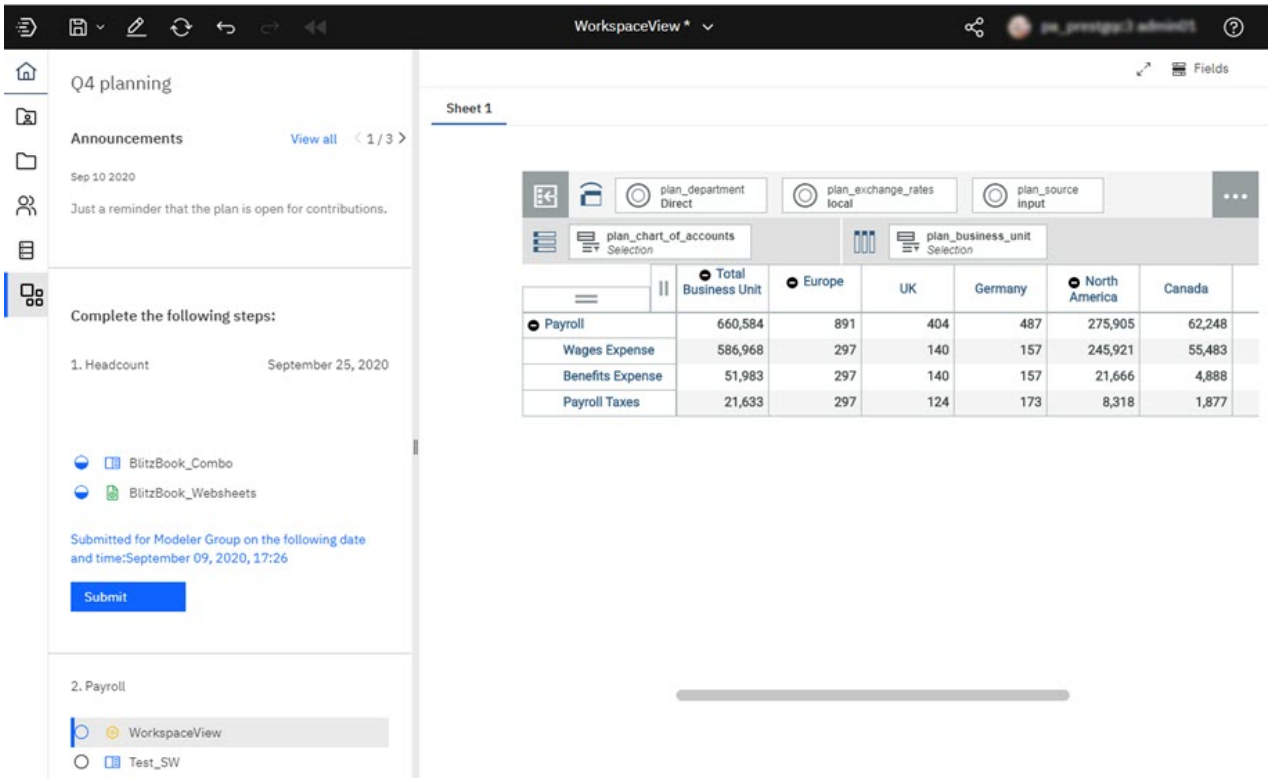
## Applications and Plans

Applications and plans let you organize logically related Planning Analytics Workspace assets such as books, view, and websheets in containers.

An application contains related assets that are grouped in sections. These sections might reflect the structure of your organization, planning and budgeting requirements, or any other relevant grouping of assets. While an application contains logically related assets, there are no implied or required actions associated with the assets or sections in an application. An individual asset can belong to more than one application.




A plan contains assets that are grouped in steps. These steps can represent discrete tasks or contributions that must be completed in a planning or budgeting process. While steps can be ordered in a plan, there is no requirement for contributors to complete the steps sequentially; they can be completed in any order. Steps can also be assigned a due date for contributions. An administrator can require that steps be explicitly submitted for approval, and an administrator can reject and reset a submission.

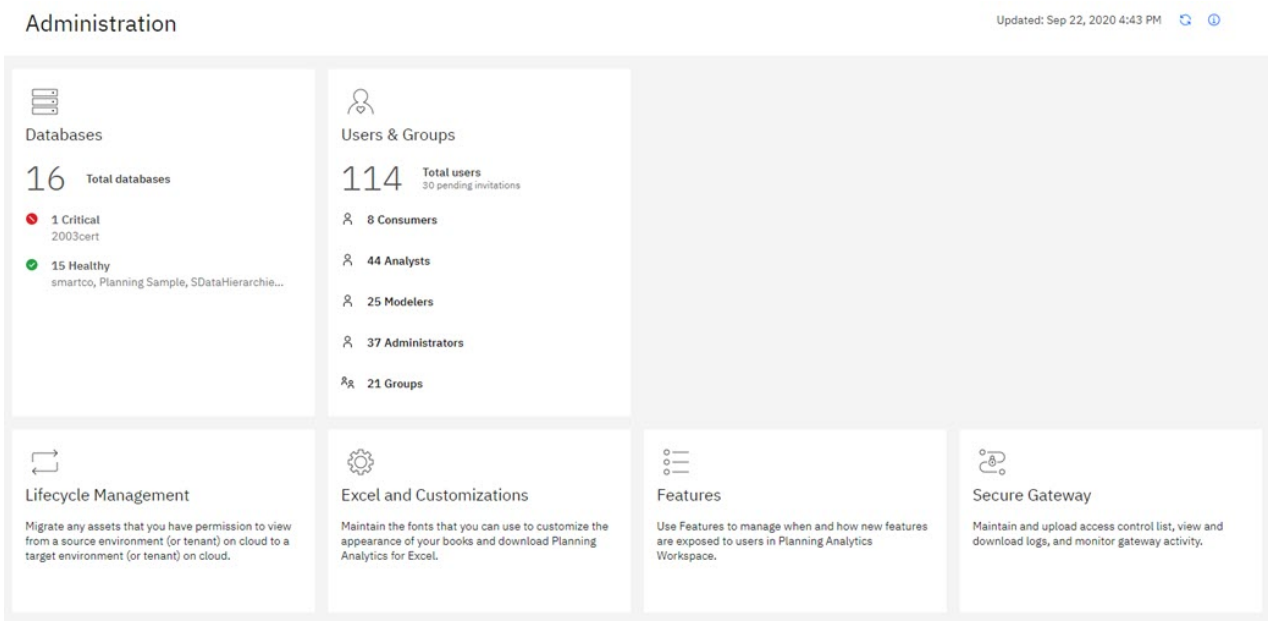



For complete details on creating, managing, and using applications and plans, see Applications and plans.

## Administration page changes

The Planning Analytics Workspace **Administration** page has been reorganized to provide greater insight into your environment and to simplify access to administrative tasks.

The Administration page now includes several task-specific tiles. The Databases tile provide quick insight into the health of your databases. The Users and Groups tile lets you know how many users are assigned to each role and the number of groups that are defined.




You can click any tile to perform the administrative tasks associated with the tile.

The tiles available on the **Administration** page vary depending on whether you are running Planning Analytics Workspace Local or on Cloud. The Agents tile is available only on Local, while the Secure Gateway tile is available only on Cloud.

For complete details on using the new Administration page, see Administer Planning Analytics Workspace.

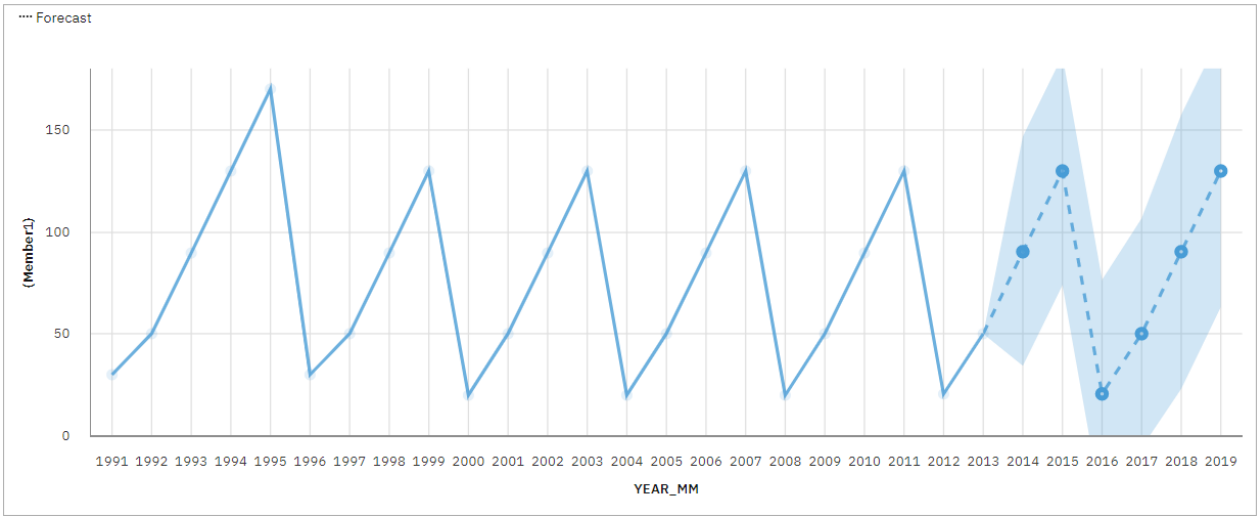
## Forecasting

You can now use forecasting in IBM Planning Analytics Workspace to discover and model trend, seasonality, and time dependence in data.

You can forecast in Planning Analytics Workspace by using automated tools that model time-dependent data. Automated model selection and tuning makes forecasting easy to use, even if you are not familiar with time series modeling.

Forecasts and their confidence bounds are displayed in visualizations as a continuation of historic data. You can also view the statistical details for generated models if you want to see the technical background.

The following example shows forecasting values and confidence bounds in a line visualization.




Specifying time series in forecasts often requires data manipulation. Planning Analytics Workspace supports a wide range of time series without the need for manipulation, ranging from standard date and time types, to nested periodic and cyclical time fields. When data is recognized as a time series, data preparation is automated. Appropriate trend and seasonal periods are detected, and models are selected from a set of nine different model types.

You can forecast in line visualizations. Forecasting allows analysis of hundreds of time series per visualization. Forecasts and confidence bounds are computed for each time series, and displayed in the visualization as extensions of the current data. You can inspect each time series separately, and tailor the forecast and results to your own data and requirements.

If you are familiar with forecasting models, you can view the selected model type, estimated model parameters, standard accuracy measures, and processing summary information.

For complete details on using forecasting in Planning Analytics Workspace, including a complete tutorial, see Forecasting.

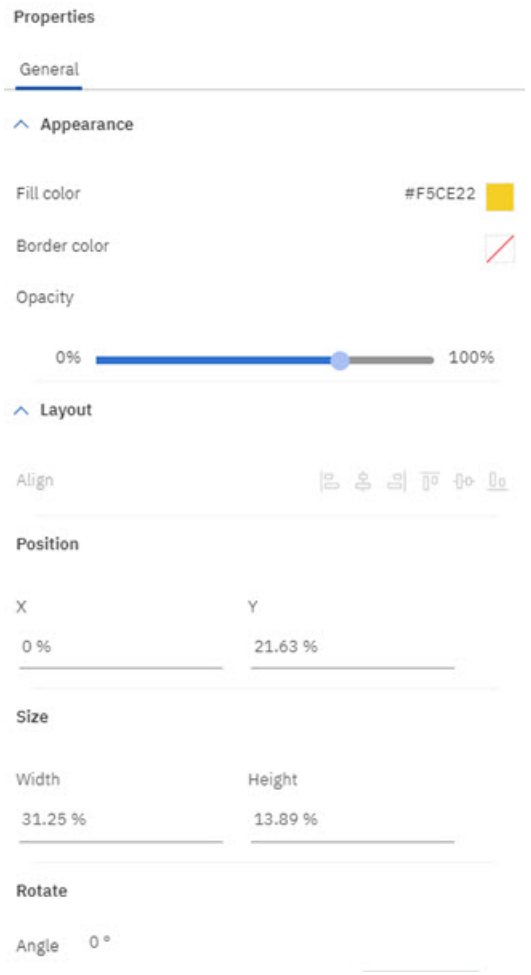
# Book and visualization improvements

The properties available to manage all aspects of your books have been significantly expanded. The visualization types available to use in Planning Analytics Workspace have been updated to provide more and improved visualizations.

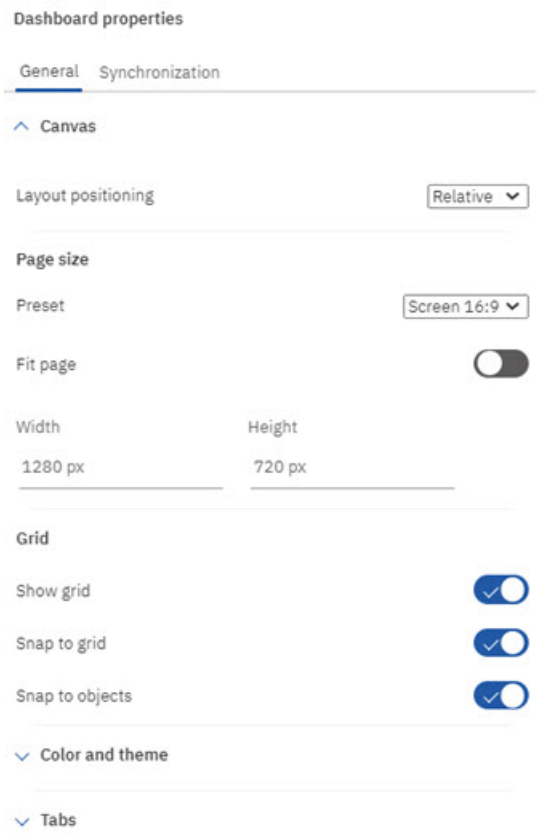
## Books

All objects in a book, such as explorations, visualizations, buttons, images, and text, can now be precisely managed using an expanded selection of properties.

When you select an object in a book and then click the **Properties** tab, you'll see an expanded list of properties that you can set to manage the object in your book. The properties available vary by object type, and the properties you're familiar with are still available, but new properties allow you to precisely manage the size, position, alignment, and appearance of all objects in a book.



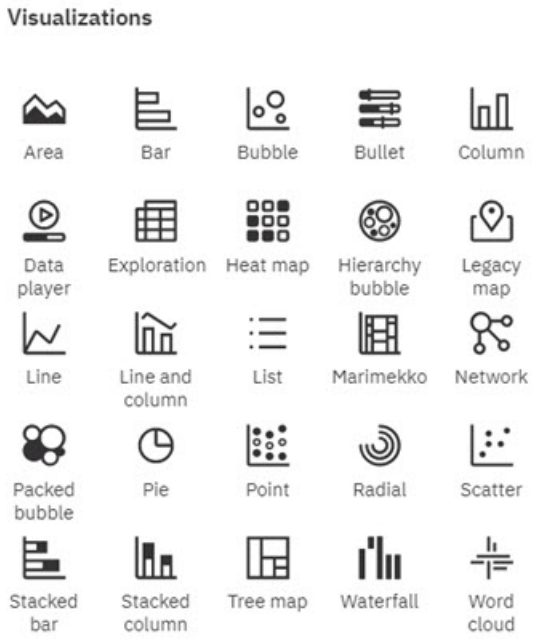

You can also set Dashboard properties to manage the general appearance of your book.



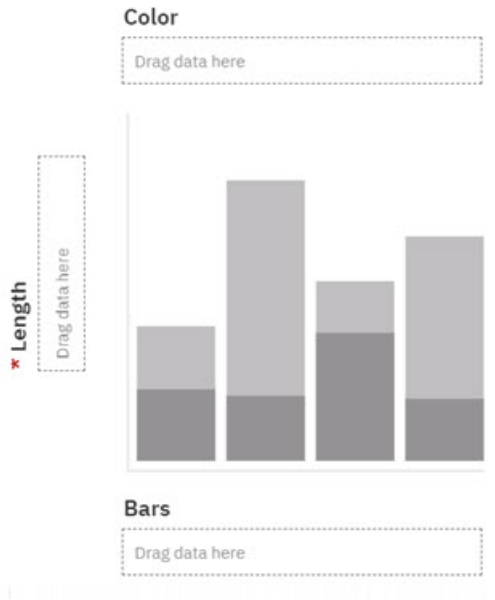

Details on using these properties are provided in the individual topics that describe how to build and manage books. For more information, see Work in books and views.

## Visualizations

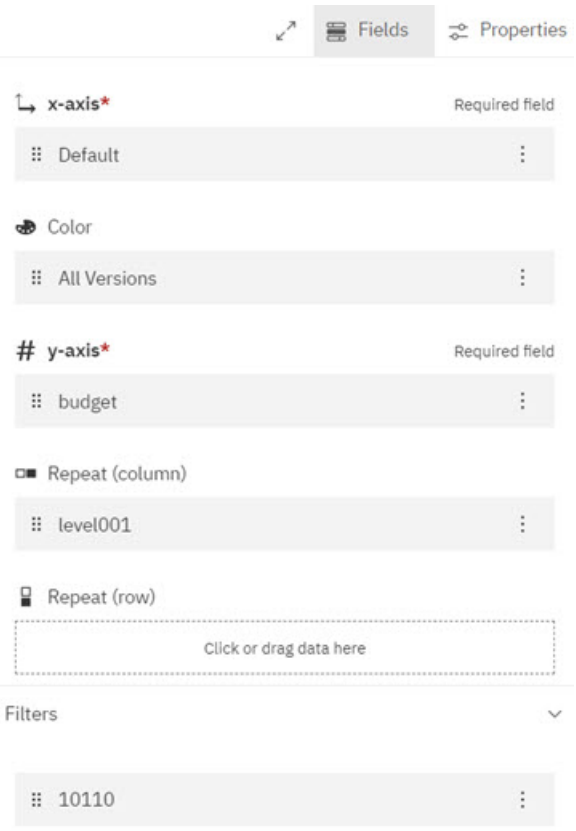
The list of visualizations available in Planning Analytics Workspace has been expanded to include twenty five options.



Explorations can be changed to visualizations in the same way you've always done it, but you can also place an empty visualization onto a book and use drop zones to build the visualization.




You can drag and drop dimensions from the Data tree onto the **Drag data here** drop zones to build a visualization from scratch. Once the visualization is complete, you can manage the visualization using the Fields tab on a book. You can drag and drop dimensions to different fields to change the structure of your visualization or you can click a field to select a new member to use in your visualization.

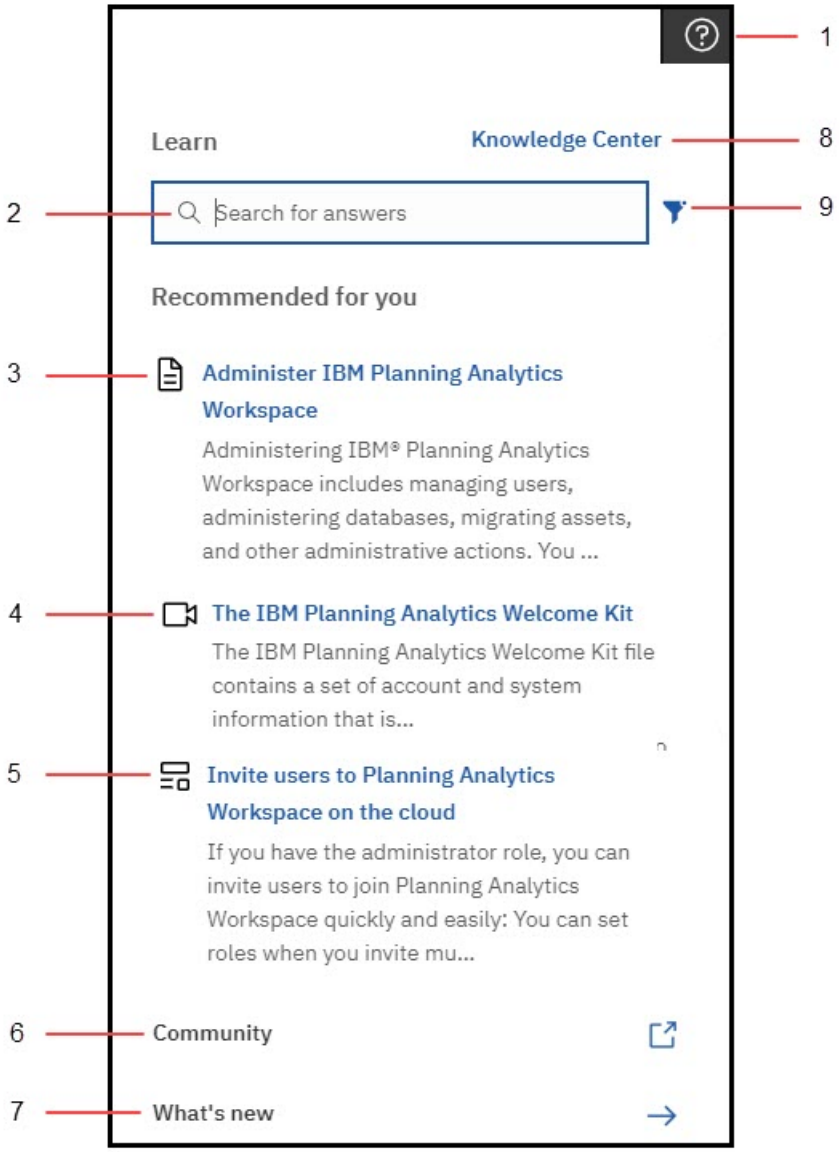



For further details, see Visualizations.

# Ask for help in the cognitive Learn pane

If you want to learn more about how to use Planning Analytics Workspace, click the Help icon and ask a question. The cognitive help tailors your help experience based on where you are working in the product and finds only the answers that are relevant to your user role. You can find the latest videos, blogs, and documentation.

Use these features on the Learn pane to find answers and be more productive!




**1**

Click the Help icon to open the Learn pane (it remembers where you were the last time you opened it). Click anywhere to close it. The Learn pane recommends content that relates to your task and finds similar content that you might also like. And, it is always learning! When you search and find answers, you are training the Learn pane and those answers influence future recommendations.

**2**

Type a question in the Search box. You can search in any supported language in the Learn pane and you see translated documentation in your search results. You also see blogs and videos that match your search, however, blogs and videos aren't translated.

**3**

Read the formal product documentation, sourced from IBM Knowledge Center.

4. Watch a video! Sometimes the best way to learn is to see it in action.
5. Read a post in the Planning Analytics Community Blog The community blog posts are written by experts who use Planning Analytics Workspace and share their tips and tricks.
6. Go to the Planning Analytics Community. In the community, you can find the latest articles, blog posts, and events. You can also start and contribute to discussions about Planning Analytics.
7. Click **What's New** to find out what is new in the latest release of Planning Analytics Workspace.
8. Visit the IBM Knowledge Center for all IBM Planning Analytics documentation In the IBM Knowledge Center, you can read all documentation, including related products.
9. Filter your search results to show only your preferred content type: videos, blogs, or documentation.

# Considerations for upgrading to Planning Analytics Workspace new experience

Users should be aware of the following considerations before upgrading to the new experience in Planning Analytics Workspace 2.0.57 SC local and 2.0.58 cloud.

The upgrade to Planning Analytics Workspace 2.0.57 SC local is a permanent upgrade. There is no way to revert to a prior version of Planning Analytics Workspace Classic.

likewise, when an administrator commits to an upgrade in 2.0.58 cloud, there is no way to revert to the 'classic' user experience.

## Features not supported in initial 2.0.57 local/2.0.58 cloud releases

These features are not supported in the initial new experience Planning Analytics Workspace release. Support for these features may be reintroduced in subsequent releases.

- Tree map visualizations upgrade successfully *except* when more than one dimension is present on an axis, in which case the tree map is converted to an Exploration during book upgrade.
- The Reports and Analysis landing page does not display individual tiles for books, views, or websheets. Rather, the Reports and Analysis landing page displays a searchable and sortable list of all assets.
- Mobile devices are not fully supported in the initial new experience release, as some gestures are not yet implemented.
- An Administrator cannot set up a global color palette. (This capability is supported as of Planning Analytics Workspace 2.0.59.)
- When you click **Share** > **Export**, you cannot share a book or view as an image or PowerPoint document. You can, however, export as PDF with enhanced print options.
- The Intent bar (sometimes called the NLP bar) is not available in this release.

## Differences in behavior between Planning Analytics Workspace 2.0.57 local/2.0.58 cloud and Planning Analytics Workspace Classic

- Chats are deprecated, as previously announced in this deprecation notice.
- Bookmarks and history are no longer available on the Data tree. Instead, you can use the Recents or Favorites tabs on the Planning Analytics Workspace Home page to open assets that you have recently viewed or favorited.
- Collections has been changed to Pins. You can pin a view or websheet from a book. You can access pinned items from the Pin button while in Edit mode.

# 2.0.55 - What's new, August 11, 2020

This version of Planning Analytics Workspace on cloud includes fixes only.

Updates to each version of Planning Analytics Workspace are cumulative. To see what was new in the previous release, see “2.0.54 - What's new, July 15, 2020” on page 408.

# 2.0.54 - What's new, July 15, 2020

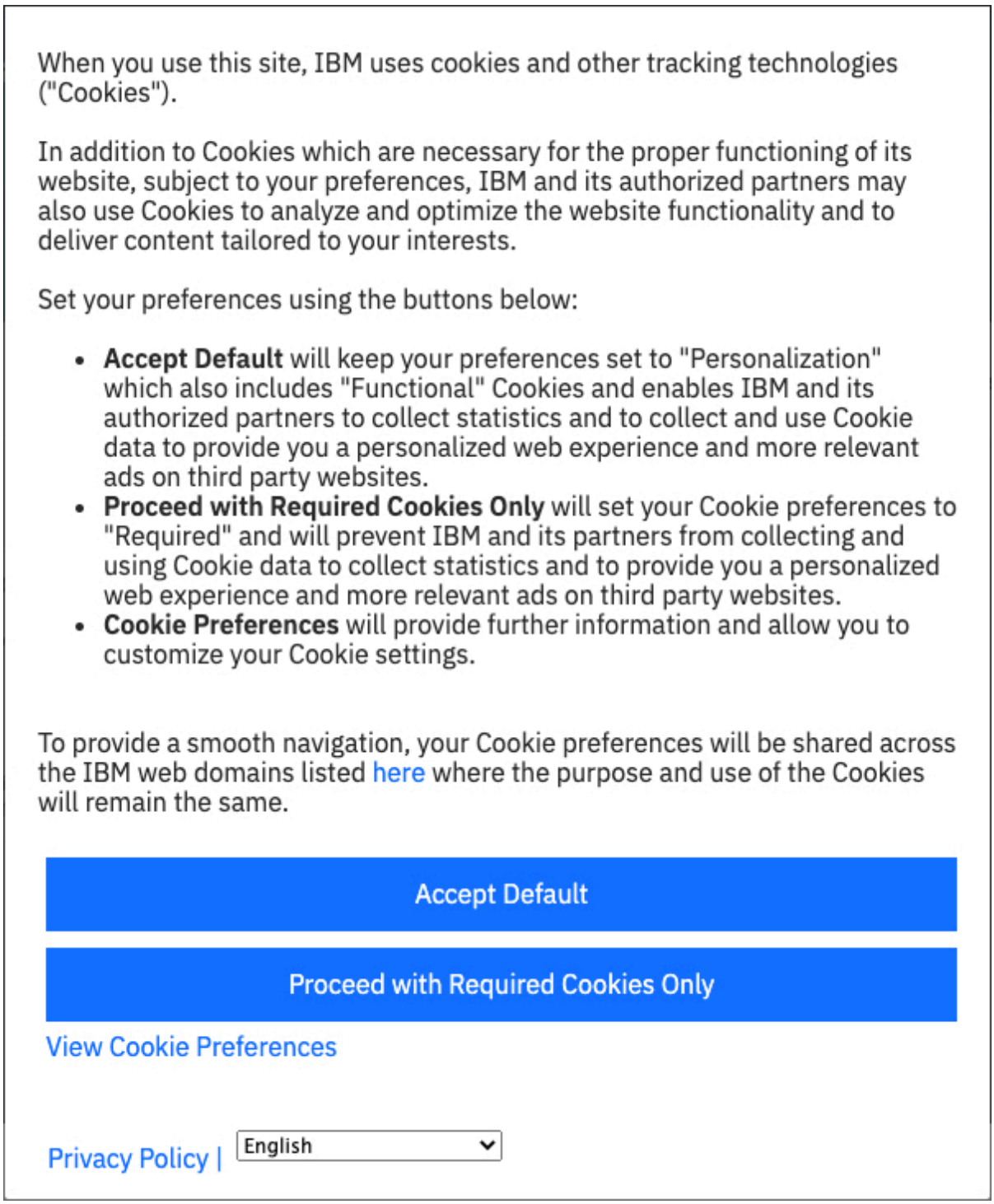
See what's new in version 2.0.54 of IBM Planning Analytics Workspace in the following topics.

**Note:** Planning Analytics Workspace on cloud is available from July 15, 2020. Planning Analytics Workspace local is available starting from July 12, 2020.

## Set your cookie preferences

IBM begins to capture metrics of your usage of IBM Planning Analytics Workspace on cloud to improve the product, and its capabilities, with the release of 2.0.54.

If you are based in the European Union, you will see a prompt the first time that you log on after Planning Analytics Workspace on cloud is upgraded. Users are determined to be in the European Union based on their IP address by using geolocation services. Your acceptance of cookie preferences is stored as a cookie in your browser, so if you clear your cookies or change browser, you see the prompt again. This is standard behavior, and enables you to control the level of cookies that you allow to be stored.




If you are not based in the European Union, you are not prompted for cookies but you can select the level of cookies by clicking and then selecting **Cookie Preferences**. You can see more information about the levels of cookies by clicking **View Cookie Preferences** in the **Cookies** window.

To learn more, see:

Cookie preferences for Planning Analytics on cloud (https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_prism_gs.2.0.0.doc/c_paw_cookies.html)

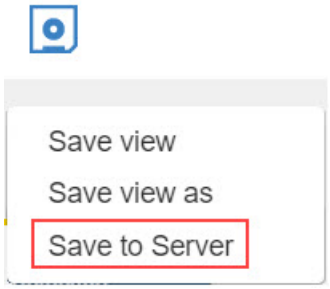
## Save a view to the TM1 database

You can save a view from Planning Analytics Workspace to the TM1 database.

When a view is saved to the database, it can be used by TM1 processes as a data source from which you can extract data and create or update objects or data.

A view saved to the TM1 database is also available to any Planning Analytics client that connects to the database.

To save a view to the TM1 database, select **Save to Server** from the shortcut bar.




To learn more, see:

Save a view (https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_prism_gs.2.0.0.doc/t_paw_save_view.html).

## Format numbers for an entire view

You can apply number formatting for an entire view from the shortcut bar.

To apply formatting, click the **Format** icon on the shortcut bar and select the desired format.



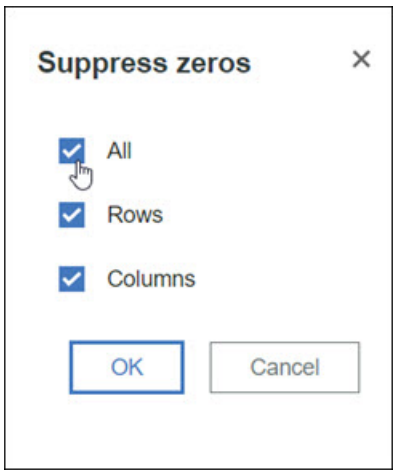

To learn more, see:

Change the format of data in a view (https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_prism_gs.2.0.0.doc/t_paw_format_data.html).

## Suppress zeros for an entire view

You can suppress zeros for an entire view from the shortcut bar.

To suppress all rows and columns that contain only zeros, click on the shortcut bar, then select **All**.




To learn more, see:

Suppress zeros (https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_prism_gs.2.0.0.doc/t_prism_suppress.html).

## Remove a user from all environments simultaneously (cloud only)

As of Planning Analytics 2.0.54, removing a user from the primary environment automatically removes the user from all environments within the organization. You no longer need to remove a user from secondary environments before removing the user from the primary environment.

# Chapter 2. Get started with Planning Analytics Workspace

The IBM Planning Analytics Workspace software is accessed in the cloud and as part of the IBM Planning Analytics Local software on premises.

For Planning Analytics Workspace in the cloud, log in with the credentials that are provided by your administrator.

To install Planning Analytics Workspace locally, see "Planning Analytics Workspace installation" in the *Planning Analytics Local Installation and Configuration* documentation on IBM Knowledge Center.

## Known issues

The following are known issues in Planning Analytics Workspace. These issues will be resolved in a future release.

### TurboIntegrator process connection to ODBC source fails in Planning Analytics on Red Hat Enterprise Linux 8 or 9

In Planning Analytics installed on Red Hat Enterprise Linux 8 or 9, a process that uses an ODBC data source fails with the error message `SQLState: 01000 SQLMsg: [unixODBC][Driver Manager]Can't open lib '/usr/lib64/psqlodbc.so' : file not found.`

To resolve this issue, add the line `LD_PRELOAD=/lib64/libssl.so.1.1:/lib64/libcrypto.so.1.1` to the `.profile` or `.bashrc` file in your Linux environment, then restart the Planning Analytics service.

### TurboIntegrator process comments starting with #Section or #Region should be modified before upgrading from Perspectives/Architect to Planning Analytics Workspace

TurboIntegrator process comments starting with `#Section` or `#Region` can cause issues in the Planning Analytics Workspace process editor.

Issues occur because `#Section` and `#Region` are reserved keywords in processes. If your TurboIntegrator process script includes comments starting with `#Section` or `#Region`, you should modify such comments before attempting to open the process in the Planning Analytics Workspace process editor.

The easiest way to modify an offending `#Section` comment is to insert a space between the # character and the first letter. `#Section` can cause issues in the editor; `# Section` does not.

`#Region` is a slightly different case. You can use `#Region` with a corresponding `#EndRegion` to create a collapsible and expandable region of code in the process editor. For example:

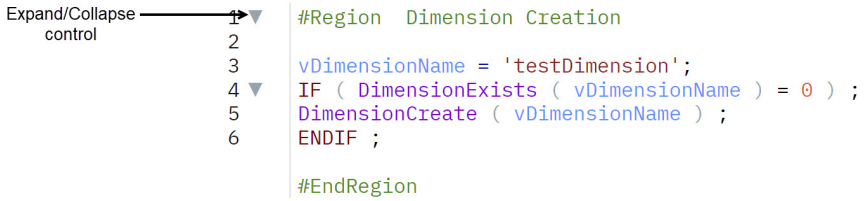
```
#Region  Dimension Creation

vDimensionName = 'testDimension';
IF ( DimensionExists ( vDimensionName ) = 0 ) ;
DimensionCreate ( vDimensionName ) ;
ENDIF ;

#EndRegion
```

This example results in a block of code that can be expanded and collapsed by clicking the Expand/Collapse control.

© Copyright IBM Corp. 2009, 2025




```
#Region  Dimension Creation

vDimensionName = 'testDimension';
IF ( DimensionExists ( vDimensionName ) = 0 ) ;
DimensionCreate ( vDimensionName ) ;
ENDIF ;

#EndRegion
```

If you want to use #Region to create an expandable/collapsible block of code, you *must* ensure that #EndRegion is present at the end of the code block.

If you want to use #Region as a general comment, you must insert a space between the # character and the first letter. #  Region is acceptable as the beginning of a comment and does not require that a closing #EndRegion be present.

# Planning Analytics Workspace local/distributed fails to start when IPv6 is disabled on Linux using kernel boot parameter

## Condition

Planning Analytics Workspace local/distributed version 2.0.75 and later fails to start when IPv6 is disabled using Linux kernel boot parameter `ipv6.disable=1`. This problem only applies to Docker Swarm deployments of Planning Analytics Workspace.

## Cause

Planning Analytics Workspace distributed version 2.0.75 and later includes a version of MySQL database that requires the kernel IPv6 support to be enabled to start up. See https://bugs.mysql.com/bug.php?id=106485 for a description of this issue.

On Linux systems where the kernel IPv6 support is disabled, some of the PAW mysql-cluster containers enter a restart loop. Inspecting the logs for the mysql-mgmt pod/container reveals an error as follows:

```
MySQL Cluster Management Server mysql-8.0.28 ndb-8.0.28
2022-04-12 09:42:01 [MgmtSrvr] ERROR -- At least one hostname in the configuration does not
match a local interface. Failed to bind on mysql-mgmt
2022-04-12 09:42:01 [MgmtSrvr] ERROR -- The hostname this node should have according to the
configuration does not match a local interface.
Attempt to bind 'mysql-mgmt' failed with error: 97 'Address family not supported by protocol'
```

You can check the host boot parameters to confirm that kernel IPv6 support is disabled:

```
$ cat /proc/cmdline
...
BOOT_IMAGE=(hd0,msdos1)/vmlinuz-4.18.0-305.19.1.el8_4.x86_64 root=/dev/mapper/rhel-root ro
             resume=/dev/mapper/rhel-swap rd.lvm.lv=rhel/root rd.lvm.lv=rhel/swap rhgb quiet
--->>>>      ipv6.disable=1         <<<<---
             noibrs noibpb nopti nospectre_v2 nospectre_v1 l1tf=off nospec_store_bypass_disable
no_stf_barrier mds=off mitigations=off
```

## Remedy

To work around this problem, kernel IPv6 support must be enabled on all the cluster nodes.

These general steps describe one possible solution to enable IPv6 in the kernel. Specific steps for resolution may vary in your environment.

1. Edit `/etc/default/grub` and remove **`ipv6.disable=1`** in line `GRUB_CMDLINE_LINUX`. For example:

   ```
   GRUB_TIMEOUT=5
   GRUB_DISTRIBUTOR="$(sed 's, release .*$,,g' /etc/system-release)"
   GRUB_DEFAULT=saved
   GRUB_DISABLE_SUBMENU=true
   GRUB_TERMINAL_OUTPUT="console"
   ```

```
GRUB_CMDLINE_LINUX="resume=/dev/mapper/rhel-swap rd.lvm.lv=rhel/root rd.lvm.lv=rhel/swap
rhgb quiet ipv6.disable=1 noibrs noibpb nopti nospectre_v2 nospectre_v1 l1tf=off
nospec_store_bypass_disable no_stf_barrier mds=off mitigations=off"
GRUB_DISABLE_RECOVERY="true"
GRUB_ENABLE_BLSCFG=true
```

2. Regenerate the GRUB configuration file.

```
$ sudo grub2-mkconfig -o /boot/grub2/grub.cfg
```

3. Reboot.

```
$ sudo shutdown -r now
```

4. (Optional) If you still want to disable IPv6, use `sysctl` settings instead of disabling IPv6 with the kernel settings.

```
$ sudo sysctl -w net.ipv6.conf.all.disable_ipv6=1
$ sudo sysctl -w net.ipv6.conf.default.disable_ipv6=1
```

5. Restart Docker.

```
$ sudo systemctl restart docker
```

# User roles

The following roles are available in Planning Analytics Workspace. The default role is **Analyst**.

**Consumer**

Consumers can open books and views and other content that is shared with them.

Consumers cannot create their own books and views, but they can share content that is shared with them, with consumer rights only.

Consumers can delete books and views if they have **Full control** permission set for those books and views.

**Note:** When you log in to Planning Analytics Workspace from an iPad, you are always a Consumer, regardless of your actual role. For more information, see “Accessing Planning Analytics Workspace from Apple iPad” on page 427.

**Analyst**

Analysts have the rights of a consumer, plus the following abilities:

Analysts can create, edit, and share books and views.

Analysts can delete books and views if they have **Full control** permission set for those books and views.

**Modeler**

Modelers have the rights of an analyst plus the following abilities:

They can create and share content, and edit content that is shared with them.

Modelers can create, edit, and delete dimensions and hierarchies. They can add, delete, cut, paste, copy, move, and sort members and their attributes in a hierarchy.

For more information, see “The modeler role” on page 718.

**Administrator**

Administrators have all the rights of a modeler, plus the following abilities:

Administrators can see all content in the workspace.

Administrators can set permissions for a book.

**Note:** At least one user in your organization must have the administrator role. If your organization has only one administrator, this user cannot be deleted or assigned to another role. Administrators can assign roles to users and only an administrator can change another user's role to administrator.

By default, the administrator role is assigned to the first user in your organization's Planning Analytics Workspace account. In Planning Analytics Workspace on cloud, this first administrator is also known as the *subscription administrator*. Only the subscription administrator can add new users.

# Manage settings for components and features

Users with the Administrator, Modeler, or Analyst role can use the Settings editor to define global settings and preferences for multiple Planning Analytics Workspace components and features.

## About this task

The Settings editor is not available for users with the Consumer role.

The components and features available in the Settings editor vary according to your user role in Planning Analytics Workspace. You'll see only the components and features for which you can set preferences.

While the Settings editor defines global settings for components and features, you can still modify settings directly in an instance of a component. For example, when using the dimension editor, you can click the **Settings** icon and modify the settings for that instance of the editor.

## Procedure

1. Click your user name on the Planning Analytics Workspace title bar.
2. Click **Profile and settings**.
3. Click **Settings**.
4. Click a settings category, then define your settings.

| Category | Setting | Description | Default value | Applicable user roles |
|---|---|---|---|---|
| **Dashboard** | Global default template | Sets the global default book template for all users when creating a new book with **New** > **Book** . | None | Administrator |
| | Personal default template | Sets your personal default book template when creating a new book with **New** > **Book** . If an administrator has defined a global default template, your personal default template takes precedence. | None | Administrator, Modeler, Analyst |
| **Database tree** | Display members | | On | Administrator, Modeler, Analyst |
| | Display object names on hover | Display the full object name when you hover on an item on the **Databases** tree. | Off | Administrator, Modeler, Analyst |

| Category | Setting | Description | Default value | Applicable user roles |
|---|---|---|---|---|
| | Hidden databases | Defines the databases that are hidden in the Databases tree. Pick the databases you want to hide from the **Select databases** menu. | None | Administrator, Modeler, Analyst |
| **Dimension editor** | Show dimension editor warning | Shows or hides the warning that appears when you open the dimension editor. The warning indicates that all edits are applied immediately when editing a dimension. | On | Administrator, Modeler, Analyst |
| | Compact header | Determines whether the dimension editor shows a compact header or a full header. When this setting is enabled, the dimension editor opens using the compact header, where the **Hierarchy** and **Filter** menus are hidden. You can click **Expand header** on the dimension editor to reveal the full header. | Off | Administrator, Modeler, Analyst |
| | Value bar | Shows or hides the value bar, which lets you edit member names, attribute names, or cell values outside of the dimension editor grid. This is helpful, for instance, when a cell is narrow but an alias name is long. | On | Administrator, Modeler, Analyst |

| Category | Setting | Description | Default value | Applicable user roles |
|---|---|---|---|---|
| | Attribute type | Shows or hides the attribute type icons on the dimension editor. Enable this option to display an attribute type icon for each attribute on the Attributes tab. Disable the option to display only attribute names on the **Attributes** tab. When the option is enabled, you'll see these icons for Alias, Number, and String icons.   | Off | Administrator, Modeler, Analyst |
| | Element index | Shows or hides the index values for dimension elements. When you enable this option, the dimension editor includes an **Index** column. | Off | Administrator, Modeler, Analyst |
| | Context menu | Shows or hides the right-click context menu for headers and cells in the dimension editor. | On | Administrator, Modeler, Analyst |
| | Page size | Sets the number of members to initially load when the dimension editor is opened and each time a new page is fetched. | 200 | Administrator, Modeler, Analyst |
| **Cube viewer** | Show hierarchy name | Shows or hides hierarchy names in the context, row header, and column header sections of the cube viewer. When you enable this option, the name of the | Off | Administrator, Modeler, Analyst |

| Category | Setting | Description | Default value | Applicable user roles |
|---|---|---|---|---|
| | | parent hierarchy appears above each member or selection name. | | |
| | Show dimension icons | Shows or hides dimension icons in the context, row header, and column header sections of the cube viewer. When you enable this option, an icon indicating the type of member or selection appears alongside the member or selection name. | On | Administrator, Modeler, Analyst |
| | Show context information | Shows or hides context information for a view. When you enable this option, the name of the view, the database it is in, and the parent cube name are displayed on the cube viewer. | Off | Administrator, Modeler, Analyst |
| | Show section headers | Shows or hides text section headers.<br>When you enable this option, the **Context**, **Row**, and **Column** headers appear for the corresponding sections. | Off | Administrator, Modeler, Analyst |
| | Show section icons | Shows or hides icons for view sections. When you enable this option, icons representing the context, row, and column sections are displayed. | Off | Administrator, Modeler, Analyst |
| **Process editor** | Font | Sets the font for the process editor. | IBM Plex Mono | Administrator, Modeler |

| Category | Setting | Description | Default value | Applicable user roles |
|---|---|---|---|---|
| | Size | Sets the font size for the process editor. | 14 | Administrator, Modeler |
| | Enable line wrapping | Enables or disables line wrapping in the process editor. | Off | Administrator, Modeler |
| | Lock generated statements | Enables or disables the ability to edit generated statements. When this option is enabled, you cannot edit any content between `#****Begin: Generated Statements***` and `#****End: Generated Statements***` in the process editor. | On | Administrator, Modeler |
| | Parameter type | Specifies the default data type for any new parameters that are added in the process editor. | Numeric | Administrator, Modeler |
| **Rules editor** | Font | Sets the font for the rules editor. | | |
| | Size | Sets the font size for the rules editor. | 14 | Administrator, Modeler |
| | Enable line wrapping | Enables or disables line wrapping in the rules editor. | Off | Administrator, Modeler |
| **Chore editor** | Use 24 hour format | When you enable this option, the chore editor uses 24 hour time format. Disable this option to use 12 hour time format. | Off | Administrator, Modeler |
| **Files and folders** | Sort by | Specifies the column that the File manager and Drive explorer use | | |

| Category | Setting | Description | Default value | Applicable user roles |
|---|---|---|---|---|
| | | to sort items in the table. | | |
| | Sort order | Specifies whether the File manager and Drive explorer display items in ascending or descending order. | | |
| **Logs** | Use 24 hour format | When you enable this option, logs use 24 hour time format. Disable this option to use 12 hour time format. | Off | Administrator, Modeler |
| | Transaction log maximum entries | | 500 | Administrator, Modeler |
| | Audit log maximum entries | | | |
| | Server log: Dark theme | Enable to apply a dark theme to the server log. | On | Administrator, Modeler |
| | Server log: Enable line wrapping | Enables or disables line wrapping in the server log. | On | Administrator, Modeler |
| | Server log: Compact header | Determines whether the server log shows a compact header or a full header. When this setting is enabled, the dimension editor opens using the compact header, where the **Log level** and **Time period** menus are hidden. You can click **Expand header** on the server log to reveal the full header. | On | Administrator, Modeler |

# Artificial intelligence capabilities in Planning Analytics Workspace

Planning Analytics Workspace includes significant artificial intelligence capabilities that help you create, analyze, and gain insight into your business data. You can learn more about these AI capabilities in the following topics.

- **"Forecasting" on page 526** - Use forecasting to discover and model trend, seasonality, and time dependence in data. Automated model selection and tuning makes forecasting easy to use, even if you are not familiar with time series modeling.
- **Planning Analytics AI assistant** - Use the Planning Analytics AI assistant to perform a wide variety of actions on books using simple natural language commands. The Planning Analytics AI assistant is available only when working directly in a book. The assistant is not available when you open a book from a plan or application, nor is it available from a modeling workbench.
- **Chart insights** - Use chart insights to view AI-generated insights in the form of text summaries and key points that help you to interpret data in a visualization.
- **"Outlier analysis" on page 474** - Use outlier analysis to identify and correct data points in an exploration that deviate significantly from the norm, potentially indicating errors or anomalies. The analysis also includes AI-generated insights for any outliers that were detected and corrected in the view, helping you to investigate anomalies and improve the quality of your data.
- **"Impact analysis" on page 476** - Use impact analysis on an exploration to determine the impact of data fields on each other as well as identify and quantify relationships across the data. Generative AI interprets and summarizes the analysis to describe the direct and indirect impact of the data fields on each other, as well as any dependencies and relationships between them.
- **Explain cell** - Use the **Explain cell** feature to get an explanation of the underlying values that contribute to a specific cell value. Cell values can be a result of complex calculations, several consolidations, or rules. The **Explain cell** feature uses generative AI to summarize the cell intersection, underlying logic of the cell value, and source type.
- **Exploration analysis** - Use the **Exploration analysis report** feature to generate an analysis of a selected view in the form of a PDF report. The report includes an AI-generated executive summary followed by an impact analysis and outlier analysis for the exploration, allowing you to take faster business decisions.
- **Data explorer** - Use the **Data explorer** feature to create views in books using natural language. The **Data explorer** feature uses generative AI to analyze your query and matches it to the relevant data in TM1 to recommend views. An explanation for the logic behind the recommendation accompanies each recommended view.

# Find information

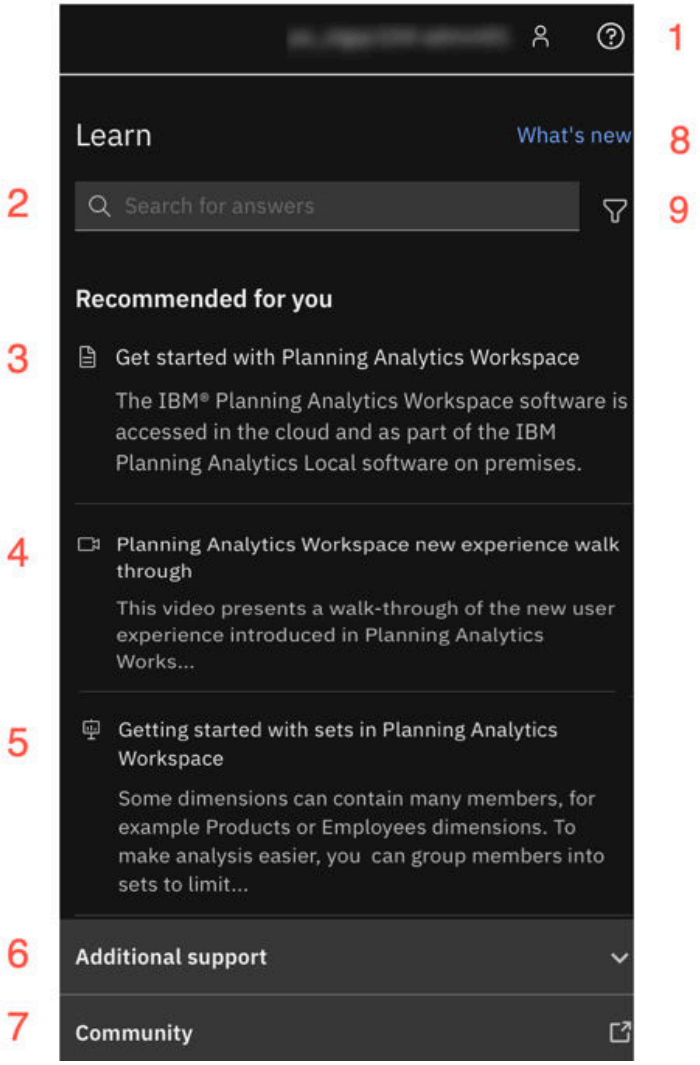
Planning Analytics documentation provides helpful information to learn and use Planning Analytics Workspace. There are unique integrated help systems, such as the **Learn Pane**, that provides you with resources that are curated to your needs. Read more about the integrated help systems Planning Analytics has to offer.

## Find information in Planning Analytics Workspace

You can view the complete set of Planning Analytics documentation on IBM Documentation (https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0).

If you want to learn more about how to use Planning Analytics Workspace, click the **Help** icon and ask a question in the cognitive **Learn pane**. The **Learn pane** tailors your help experience based on where you are working in the product and finds only the answers that are relevant to your user role. You can find the latest videos, blogs, and documentation.

Use these features on the **Learn pane** to find answers and be more productive!




**1**

Click the **Help** icon to open the **Learn pane** (it remembers where you were the last time you opened it). Click anywhere to close it. The **Learn pane** recommends content that relates to your task and finds similar content that you might also like. And, it is always learning! When you search and find answers, you are training the **Learn pane** and those answers influence future recommendations.

**2**

Type a keyword in the **Search** field. You can search in any supported language in the Learn pane and see translated documentation in your search results. You also see blogs and videos that match your search, however, blogs and videos aren't translated.

The Learn pane predicts your search query as you type keywords in the **Search** field and displays a list of search suggestions from all content types.

Not sure what to search for? Click the **Search** field to see a list of popular search suggestions.

**3**

Read the formal product documentation, sourced from IBM Documentation.

If you're using Planning Analytics Workspace 2.0.90 or later versions, you'll notice that the **Learn Pane** has dynamic and static recommendations. Dynamic recommendations are AI-generated and change on a daily basis based on what you and other users are viewing. Static recommendations remain unchanged if no content was viewed in the **Learn Pane**.

**4**

Watch a video! Sometimes the best way to learn is to see it in action.

5. Read a post in the Planning Analytics Community Blog. The community blog posts are written by experts who use Planning Analytics Workspace and share their tips and tricks.
6. Need help with an issue? Click **Additional support** to access Documentation or Support. You can also submit an idea!
7. Go to the Planning Analytics Community. In the community, you can find the latest articles, blog posts, and events. You can also start and contribute to discussions about Planning Analytics.
8. Click **What's New** to find out what is new in the latest release of Planning Analytics Workspace.
9. Filter your search results to show only your preferred content type: videos, blogs, or documentation.

# Planning Analytics Trial

IBM Planning Analytics Trial provides you with the opportunity to use and experiment with some of the components and capabilities of IBM Planning Analytics, most notably **Reports and Analysis** and **Planning Analytics for Microsoft Excel**.

There are some limitations and configuration requirements you should be aware of as a Planning Analytics Trial user.

## Planning Analytics Workspace

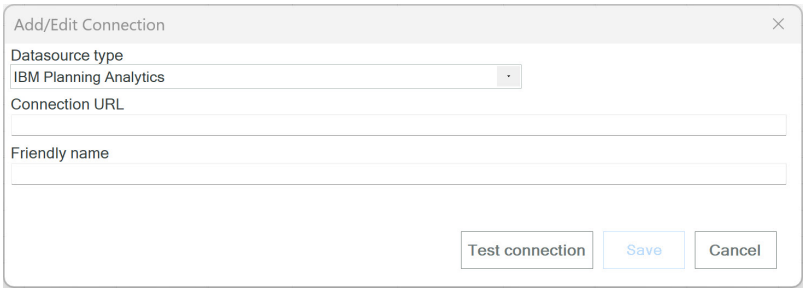
These limitations apply to Planning Analytics Trial users.

- Trial users cannot use Applications and Plans.
- Trial users cannot use the modeling workbench to create and modify model objects such as dimensions, cubes, and rules.
- Trial users can use Planning Analytics Administration to configure, monitor, and administer a Planning Analytics Workspace environment, but cannot create a new Planning Analytics database.

## Planning Analytics for Microsoft Excel

If you're using Planning Analytics Trial on Windows, you can download and install Planning Analytics for Microsoft Excel.

When you first open Planning Analytics for Microsoft Excel, you are asked to specify a **Datasource type** and a **Connection URL**.



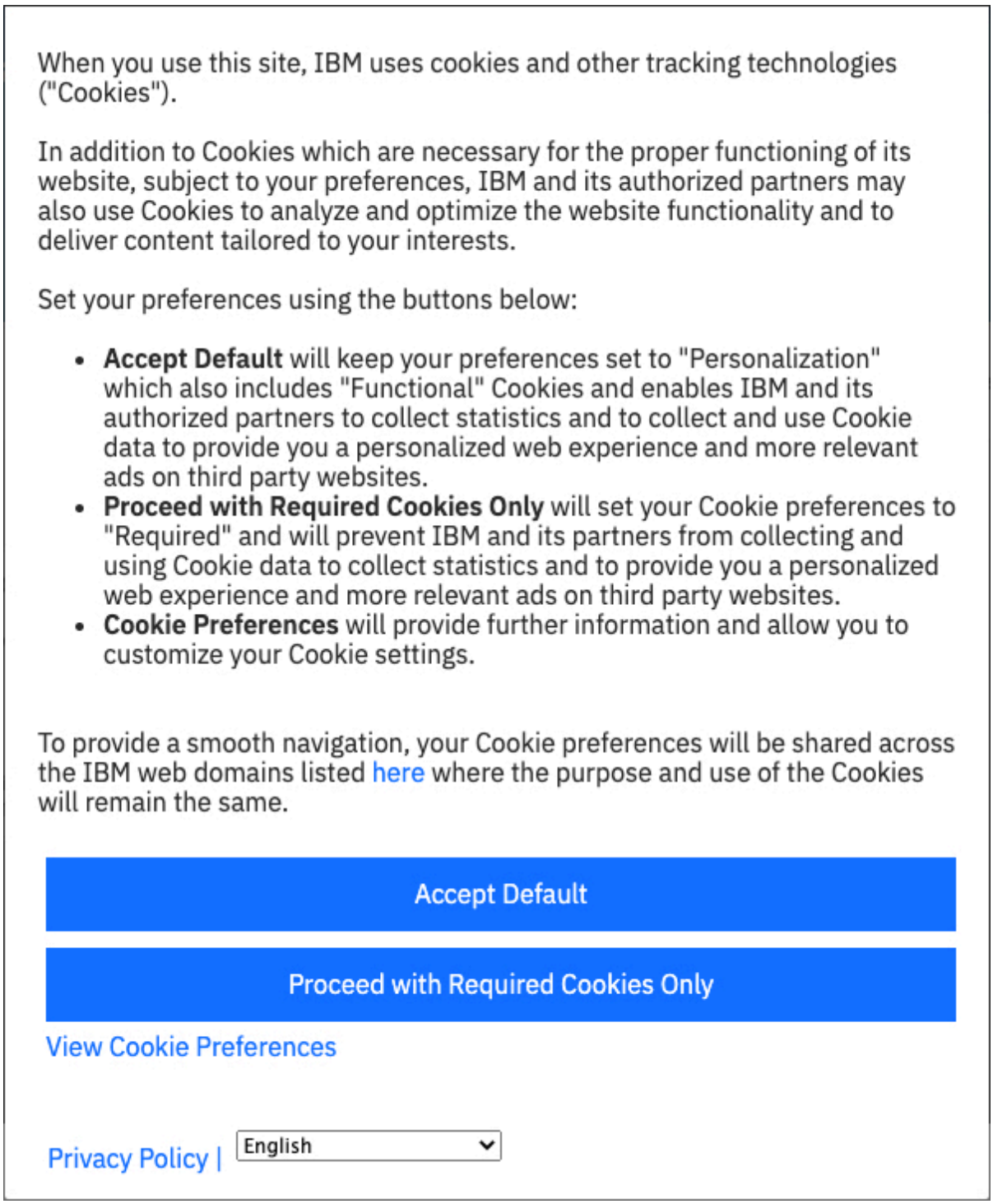

- Select **IBM Planning Analytics** as the **Datasource** type.
- The **Connection URL** should be the same URL you use to access Planning Analytics Trial. In most cases, the URL is `https://us-east-1.planninganalytics.saas.ibm.com/`.

  To confirm the URL, check the address bar on your web browser while logged in to Planning Analytics Workspace in Planning Analytics Trial.

# Cookie preferences for Planning Analytics on cloud

IBM Planning Analytics creates cookies when you go to the website, and saves them to your computer. You can customize the level of cookies that are created.

If you are based in the European Union, you will see a prompt the first time that you log on after Planning Analytics Workspace on cloud is upgraded. Users are determined to be in the European Union based on their IP address by using geolocation services. Your acceptance of cookie preferences is stored as a cookie in your browser, so if you clear your cookies or change browser, you see the prompt again. This is standard behavior, and enables you to control the level of cookies that you allow to be stored.



If you are not based in the European Union, you are not prompted for cookies but you can select the level of cookies by clicking  and then selecting **Cookie Preferences**. You can see more information about the levels of cookies by clicking **View Cookie Preferences** in the **Cookies** window.

The following options are available.

**Required Cookies**
The minimum level of cookies needed to use IBM Planning Analytics.These cookies are needed to provide basic functions as you browse IBM websites.

**Required Cookies** enables features such as:

- Cookie consent collection and cookie blocking for privacy controls.
- Secure log-in and transactions (that is, authentication, single sign-in, remember settings) across web pages.
- Online services, load balancing, and performance.
- Remembering transaction progress during the session.

**Functional Cookies**
Functional Cookies provides a better user experience than Required Cookies. The Functional Cookies selection captures and remembers user preferences in IBM websites, enhances their usability, analyzes site usage, and enables site optimization.

In addition to the **Required Cookies** features, **Functional Cookies** enables functions such as:

- Remembering your log-in details for automatic log-in across session.
- Analyzing site usage to provide better content experiences.
- Conducting analytics to optimize site functions.
- Making sure that the website looks and feels consistent.

**Personalization Cookies**
Personalization Cookies improve your overall experience of IBM websites and tailor content/ advertising to your interests.

- Allow personalization of IBM page content.
- Allow delivery of IBM marketing relevant to your interests on IBM and on third-party publishers.

# Applicability of Planning Analytics Workspace documentation

The documentation available through the integrated help system and on IBM Knowledge Center always reflects the most recent full capabilities of Planning Analytics Workspace on cloud.

## Cloud versus on-premises

In some instances, the current capabilities on cloud differ from the most recent Planning Analytics Workspace Local update. This is because local updates are made available only after cloud deployment occurs. In this case, Planning Analytics Workspace Local customers might see documentation for features that are not yet available to them.

Similarly, while Planning Analytics Workspace on cloud is refreshed on a regular cadence, Planning Analytics Workspace Local customers might choose to update their installation on a less frequent basis. This can also result in a difference between the features that are described in the documentation and the features that are available to a Planning Analytics Workspace Local customer.

## Deprecation

In some cases, the current documentation differs from what is available in your environment because functionality is deprecated in the current release of IBM Planning Analytics. In the documentation,

Deprecated indicates deprecated functionality that will be removed in a future release. Not supported indicates functionality that is no longer available or supported in the current release.

### TM1 Server

Depending on the Planning Analytics version that a Planning Analytics Workspace cloud customer uses, some features that are documented might not be available to use or might not be visible in the user interface.

### Device and browser compatibility

Some features that are documented and available in Planning Analytics Workspace on a desktop browser might not be available on Apple iPad. For more information, see "Accessing Planning Analytics Workspace from Apple iPad" on page 427.

## Accessing Planning Analytics Workspace from Apple iPad

You can access Planning Analytics Workspace from an Apple iPad. Other mobile devices are not currently supported in Planning Analytics Workspace.

When you log in to Planning Analytics Workspace from an iPad, you are always working in Consumer mode, regardless of your actual defined role assignment.

In Consumer mode you can:

- Open the Welcome page and view the folder structure
- Open existing views, books, and websheets
- Change cell formats
- Switch visualization types
- Interact with explorations and visualization (drill down, hide members, reorder dimension, and so on)
- Use selectors to set context for explorations and visualizations

In Consumer mode on a mobile device, you *cannot*:

- View or access the data tree
- Use or enable Edit mode
- Perform any administration tasks
- Save an existing view or book, or use save-as to create a copy of a view or book
- Create a view or book
- Open the Set Editor to modify existing sets or create new sets
- Create member calculations or summary calculations
- Share a book or view in any manner (download, email, or print)
- Delete or rename a book or view
- Add a view to a collection

In addition, you cannot create conditional formatting on an iPad.

### Performance considerations

When you open a view on an iPad, Planning Analytics Workspace initially loads and displays only the number of rows that fit on a single page. Depending on how the view is configured, this can be anywhere between 30 - 50 rows.

As you scroll through the view, Planning Analytics Workspace loads each subsequent page sequentially. This occurs even if you rapidly scroll through several pages without pausing for each page to be displayed.

For example, if you are looking at a value on the first page of a view, and then scroll to page 8 of the view, pages 2 - 7 must be loaded before page 8 can be displayed. There is a slight delay as each page is loaded.

To maintain minimal load times, limit views that are accessed from iPads to 120 or fewer rows.

Additionally, you can improve performance by:

- Limiting the number of views on a single sheet (the best performance is achieved with a single view on a sheet)
- Using chart visualizations in place of explorations (grids) whenever practical
- Limiting the scope of synchronization between views to the lowest possible level

## Supported languages

IBM Planning Analytics Workspace supports different languages in its user interface as well as its documentation.

The following table shows the languages supported in the user interface and in the documentation.

*Table 2. Supported languages in Planning Analytics Workspace.*

| Language | User interface | Documentation |
|---|---|---|
| Brazilian Portuguese | ✓ | ✓ |
| Danish | ✓ | ✗ |
| English | ✓ | ✓ |
| French | ✓ | ✓ |
| German | ✓ | ✓ |
| Italian | ✓ | ✓ |
| Japanese | ✓ | ✓ |
| Korean | ✗ | ✓ |
| Norwegian (Bokmål) | ✓ | ✗ |
| Polish | ✓ | ✗ |
| Russian | ✓ | ✗ |
| Simplified Chinese | ✓ | ✓ |
| Spanish | ✓ | ✓ |
| Swedish | ✓ | ✗ |
| Traditional Chinese | ✓ | ✗ |
| Ukrainian | ✓ | ✗ |

# Chapter 3. Work in books and views

Books contain related data in the form of views, websheets, scorecards, visualizations, graphics, videos, and embedded web pages. Views are defined selections of a cube that are used for analysis, exploration, and data entry.

## Books

Books can contain cube views, visualizations, scorecards, dimension selectors, graphics, videos, and embedded links to web pages.

You can do the following tasks in a book:

- Collect and arrange content to support a process.
- Add navigation and text to guide users through a process.
- Customize the formatting and layout by adding logos, videos, graphics, and applying formats and text styles.
- Include content from different cubes and databases in the same book.
- Share books and views with people.

### Create a book

Create an IBM Planning Analytics Workspace book to contain your data. Use a defined template or build your workspace with a freeform template.

#### Create a book in Planning Analytics Workspace

##### Procedure

1. In a web browser, go to the URL for IBM Planning Analytics Workspace and log in with your credentials.
2. Click **Home** 

, **New**, then click **Book from template**.
3. Select a template to use and click **Create**.

   IBM Planning Analytics Workspace provides templates that contain predefined designs and grid lines for easy arrangement and alignment of the visualizations.

   When you create a new book, choose a tabbed template to create a book with multiple tabs or choose a single page template. You can also hide tabs on a tabbed book if you want just one tab and less clutter on the screen when you switch to preview mode. For more information, see “Hiding the tabs ” on page 442.
4. Alternatively, choose **Book** to create a book without having to specify a template or layout. This option is slightly faster, and when you choose this option, the new book is created with the blank tabbed template.
5. Click the **Properties** icon  to modify book properties such as visual themes, layout positioning, and tab style options. To close the **Properties** pane, click the **Properties** icon again.
6. Save your book by clicking  and selecting whether the book is saved in the **Shared** or **Personal** folder. If you have sub-folders within **Shared** or **Personal**, you can select which folder the book is saved in.

   Once you create a book, you can save it as a default template. Administrators can also make a book template a global default for all users. If you have a default book template, it is automatically applied

© Copyright IBM Corp. 2009, 2025

to any new book you create. To create a blank book, click **Book from template** and choose the blank template.

**Important:** When naming a book in Windows, you can use any characters except for the following: / \ : * ? " < > |

## Edit mode

Switch to edit mode to modify and assemble a book.

### Edit mode in Planning Analytics Workspace

When a book is in edit mode, you can do the following tasks:

- Add new content.
- Move and resize content.
- Make sure that any changes that you make to widget settings are kept. For example, if you are not in edit mode, some actions such as pivoting and slicing are not kept across sessions.
- Rename the book.
- Use properties to configure colors, themes, layouts, and so on.
- Add new tabs to create a book with multiple tabs and to organize your data. .

Turn on the **Edit** switch 

to go into edit mode, and turn it off to leave edit mode.

## Save a book

Save your book to preserve your changes.

To save a book:

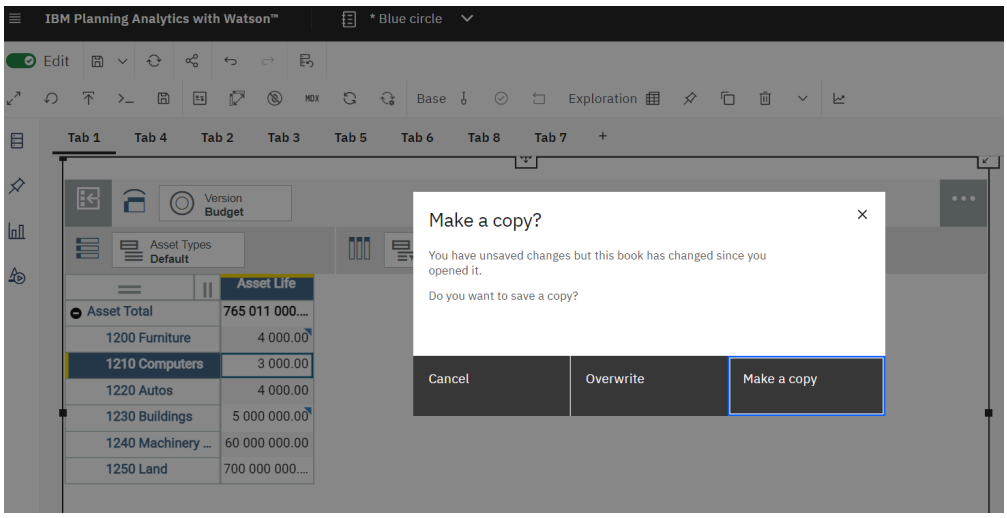
1. Click **Save** on the toolbar in the book.




2. If you are saving your book for the first time, select whether the book is saved in the **Shared** or **Personal** folder. If you have sub-folders within **Shared** or **Personal**, you can select which folder the book is saved in.

   **Note:** You can also save your book as a template. For more information, see Save a book as a template and set it as default.

When multiple users are working in the same book, the following message displays if the book was modified after you opened the book:




To save your changes, you can create a copy of the book or overwrite the changes made by another user.

## Save a book as a template and set it as default

You can save a book as a template to reuse its arrangement and avoid starting a book from scratch. You can also make it your default template. Additionally, administrators can set a book to be a global default template to make it a default for other users.

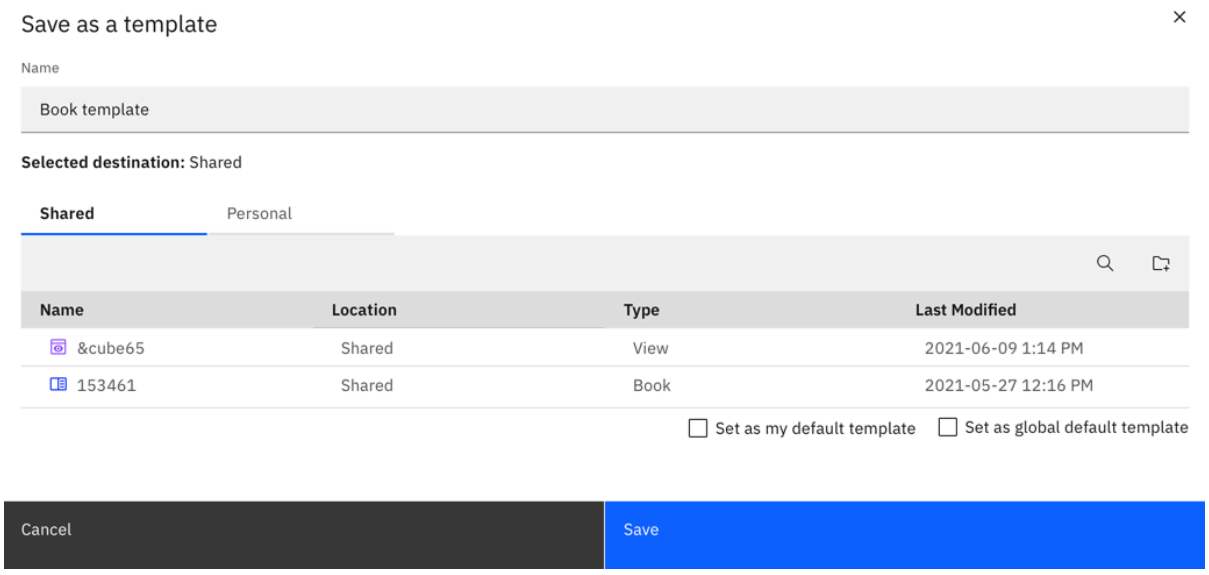
When a book template is applied, only the static data (such as properties related to widget appearance, fonts, or styling) in the template is transferred to the new book. Dynamic data that is tied to a TM1 server (such as the content of a widget, properties that are related to synchronization, or TM1 rules and processes) is removed.

For example, if your book template has a **Bar** visualization, when you apply the template to a new book, a blank **Bar** visualization is transferred to the new book. You can then add data to the **Bar** visualization from any database, giving you the flexibility to add the data you need but in a structured format.

### Saving a book as a template

To save your book as a template, follow these steps:

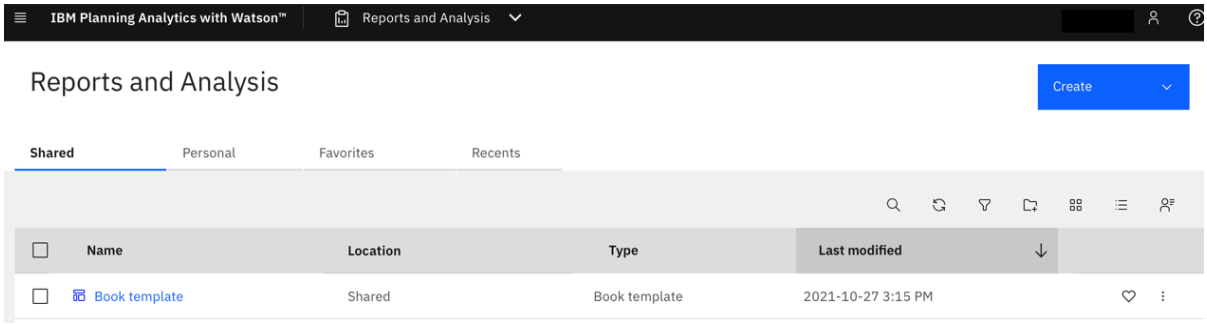
1. Click  to save your book and select **Save as template**.
2. In the **Save as template** dialog box, give your book template a name.



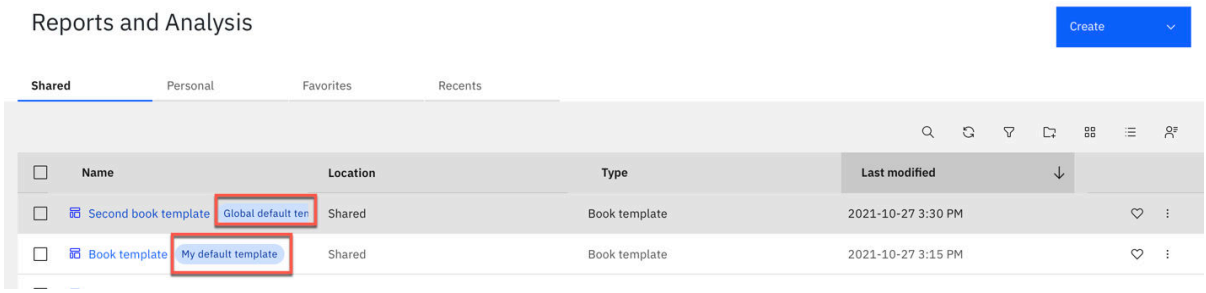
3. Select the destination folder where you want to save the template.
4. If you want to set your book as your default template, select **Set as my default template**. Administrators can also select **Set as global default template** and make it a default template for other users.

   Skip this step if you do not want to set the book as a default template; you can make it a default later.
5. Click **Save**.

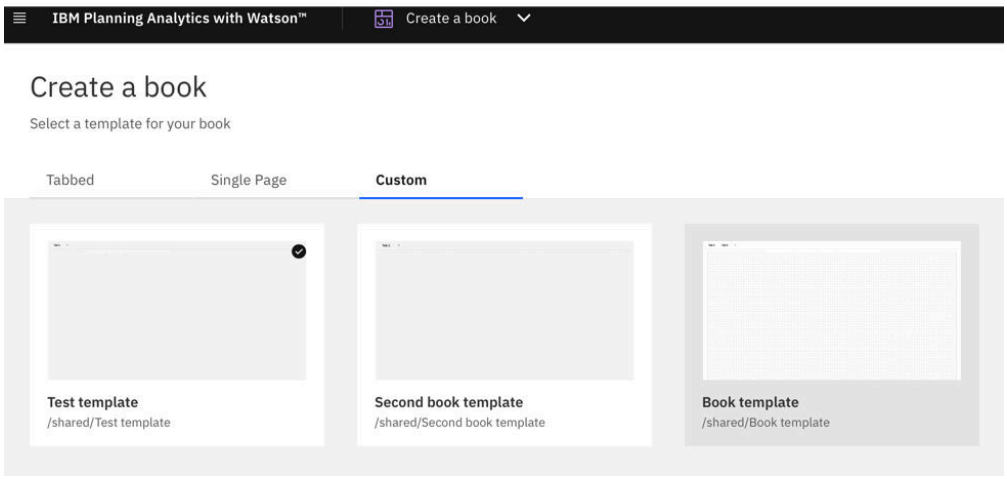
After you create your book template, it displays in your assets under **Reports and Analysis** on the **Home** page and in the destination folder in your side panel. Filter the assets to see only the book templates.




When you save a book as your default or global default template, the related tag displays next to the book name.




You can also see all your book templates under the **Custom** tab when you create a new book from a template.




In Planning Analytics Workspace 2.0.93, when you save a book as a template, any data format that you applied to an entire view or visualization also saves to the template. When you create a new book using the template, the retained data format automatically applies to the view or visualization when you populate the data. The data format saves to a template only if you applied the format to an entire view or

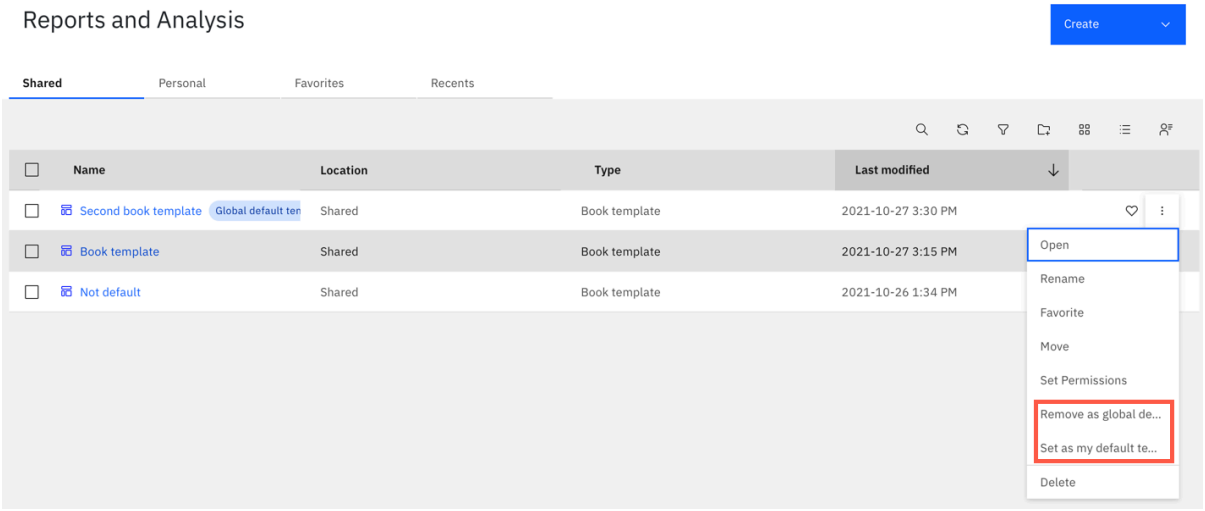
visualization by clicking **Format values** in the application toolbar. The data format is retained even when you replace the current view in the new book with another view.

## Setting or removing the default option from a book template

To set the default option on a book template or to remove it, follow these steps:

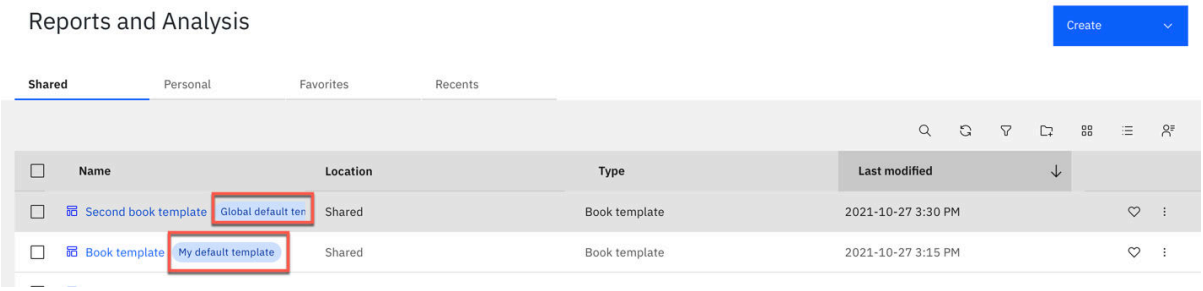
1. Access the side panel by clicking 

or go to **Reports and Analysis** on the **Home** page.
2. Select the destination folder in which you saved the book template.
3. Click the ⋮ icon for your book template to see menu options.




4. Select the option that you want for your book template.

You can have only one default and one global default template.




However, a book template can be both a user default and global default template at the same time. When a book template is both your default and global default, only the **Default template** tag displays next to the book name.

**Note:** The user default template always takes preference over the global default.

## Editing a book template

Similar to editing a book, you can edit a book template.

1. Open the book template from **Reports and Analysis** on the **Home** page or from the side panel.
2. Edit the book template and click **Save**.

## Share a book

You can share a book as a link, a PDF, or in an email.

When you share a book with a link, other users can only see the data that they have permission to see, as defined through Planning Analytics security. For example, if a view has a business unit dimension that contains 10 European business units, but a user has rights to see the data from just the UK and Germany business units, only data for the UK and Germany are visible to that user when they open the book from the link.

When you share a book as a PDF or in an email, you share a static image of the book. All data visible in the book is visible to any other person with whom you share the book.

When you set permissions for books and views, you can choose the level of access that users can have from one of the following options: **View**, **Edit**, or **Full control**.

**Note:** Email sharing and notifications must be enabled by an administrator in IBM Planning Analytics Workspace Local. For more information, see Enable email notifications in Planning Analytics Workspace Local.

### Procedure

1. Click to share a book or view.
2. Click the tab for the type of share you want to use.
   - Click **Link** to get a link that you can give to another Planning Analytics user and iframe code you can embed in an HTML page.
   - Click **Export** to export the book to PDF. You can choose the page size and orientation for the PDF.
   - Click **Email** to share the book as an email attachment.
     a. Type an email address, subject, and a message for the email recipient.
     b. Select the format that you want to use for the attachment: **Image** (PNG), **PDF**, or **PowerPoint**.
     c. If images of the book have not yet been generated, click **Generate images**. If the images have been generated, but the book has since been modified, click **Refresh**.
     d. Select the generated images that you want to attach to the email.
     e. Enable the **Add link** option to include a link to the book in the email.
     f. Click **Send**.

## Resetting a book

You can use **Reset book** to reverse your changes and go back to the last saved version of the book.

Resetting a book is useful if you want to reset all the filters of a book to their initial selections.

For example, if you changed a widget in your book, click **Reset book** to reverse your changes. Your book returns to its last saved view.




To enable the **Reset book** button, you need to first save your book.

**Note:** You cannot use **Reset book** to undo data entries. To undo a data entry, click **Undo** in the toolbar.

When you reset your book from a saved tab, the book restores to that currently open tab. If you reset a book from a tab that has not been saved, the book returns to the first tab.

# Setting up the page in a book

You can set up your book to create the layout you need for your business purposes.

## About this task

You can set whether the layout positioning is relative or absolute.

When you choose a layout style, consider the screen size of the device that will be used to view the book.

You can define the page size or select the page size from a number of presets.

If you choose absolute layout positioning, the objects retain their size and position, regardless of the screen size of the device that they are being viewed in. For small screen sizes, absolute positioning can increase the need for scrolling because objects are larger than the screen. Absolute positioning results in one large area that is not divided into sections. Choose absolute if you want visualizations to appear exactly as you size and place them, regardless of the screen size of the device that they are viewed in.

The option **Fit page** resets the view of the book to fit your screen. When you select absolute layout positioning, then the option **Fit page** is not available.

If you choose relative layout positioning, the objects adjust their size and position relative to each other, the data they show, and the screen size of the device that they are viewed in. Select relative layout positioning if you want the size and position of visualizations to adjust to fit into the screen size of the device that they are viewed in. The appearance of the book can change depending on the size of the screen that it is viewed in.

## Procedure

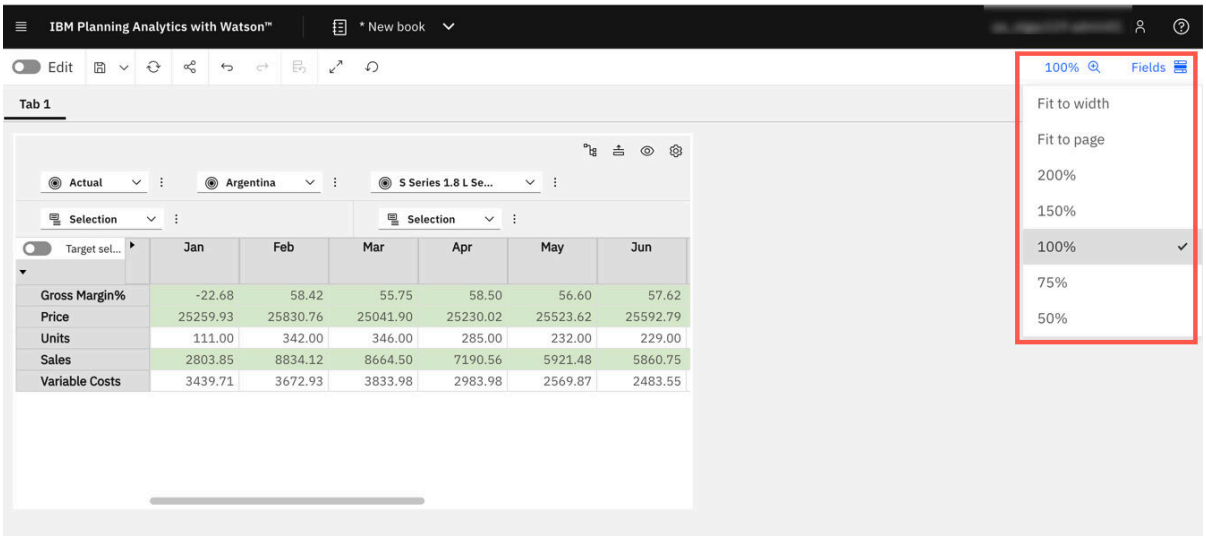
1. Open a book.
2. Click the **Properties** icon.

   If you don't see the **Properties** icon, click the **Edit or preview** icon.
3. Under **Canvas**, select a relative or absolute layout positioning.
4. Under **Page size** select the page size from the **Preset** menu or specify the width and height yourself. Optionally select the **Fit page** option.

# Adjusting page size

You can adjust the page size in books with the zoom feature. Click the zoom icon on the toolbar and select a preset size to see your content at that magnification level. You can also size the page view to fit the page or fit the width of your window. In both cases, the page size is responsive and automatically adjusts to fit the page or window when you resize your window.

**Note:** The zoom feature is available when the **Edit** mode is disabled.

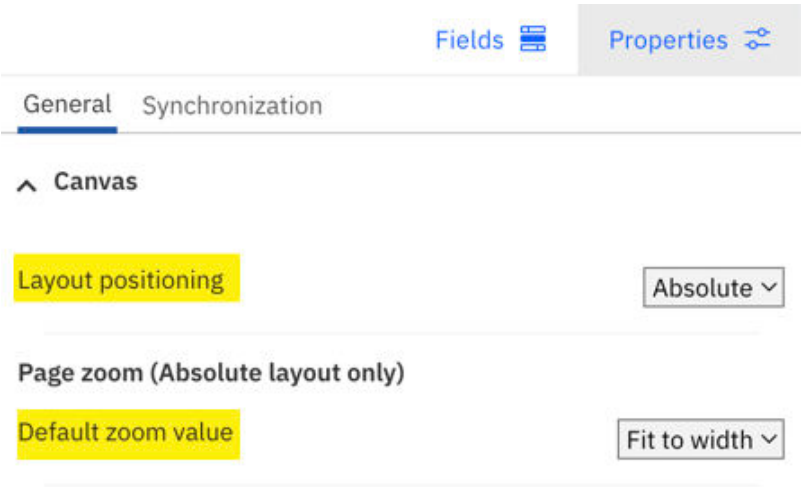



To use the zoom feature, make sure that the book is in **Absolute** layout. The zoom feature is not supported in books that use the **Relative** layout position.

To check the layout position of your book:

1. In **Edit** mode, click **Properties**.
2. Under the **General** tab, expand the **Canvas** section and make sure **Absolute** is selected for **Layout positioning**.

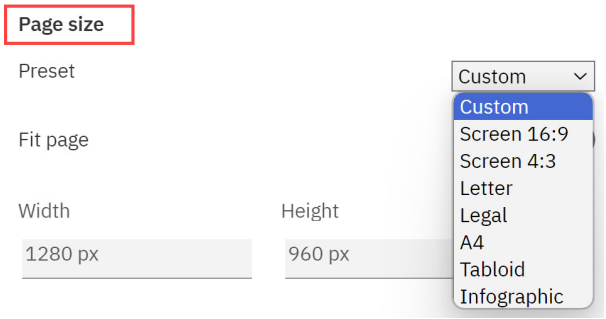
You can also set a default zoom value for your book under **Properties** so that it always opens in the selected size.




If you set a default zoom value and then enter the full screen mode, your default zoom value is overridden and the page size changes to **Fit to page** to allow you to see the full contents of your page.

Click the zoom icon to manually change your zoom value in full screen mode. When you exit full screen mode, you automatically go back to the previously selected zoom value.

You can also set a page size independent of zoom. Select one of the **Preset** sizes or select **Custom**, then set a width and height for the page.




The **Preset** options represent common page and screen sizes.

# Snapping objects to a grid and to each other

You can display a grid on your book to help you place objects exactly where you want them. You can snap objects to the grid and snap objects to each other.

## Procedure

1. Click the **Properties** icon.
2. For the entire book, ensure that nothing else has focus by clicking the background, away from any visualization or object.
3. On the **General** tab, set **Grid** options:

| Goal | Action |
|---|---|
| Show or hide the grid on the canvas | Set the **Show grid** option to on or off. This option is on by default. |
| Snap objects to the grid on the canvas | Set the **Snap to grid** option to on or off. This option is on by default. |
| Snap objects to each other | Set the **Snap to objects** option to on or off. This option is on by default. |

# Change colors

You can change colors in the entire book, in a visualization, or in widgets that you added to a book.

- For the book, you can change the visual theme, the color palette, and the background color.
- For visualizations, you can change the color palette, the color used in the elements (such as bars, bubbles, or lines), the fill color, and the border color. You can also make the visualizations more transparent or opaque.
- For shape and text widgets, you can make the widget more transparent or opaque, change the fill color, and change the border color.

## Change colors in Planning Analytics Workspace

### Procedure

1. Click the **Properties** icon.
2. For the entire book, ensure that nothing else has focus by clicking the background, away from any visualization or object.
3. Click the **General** tab, then **Color and Theme**.

| What to change | Actions |
|---|---|
| The theme of the entire book | Select a **Visual theme** for the book. |
| The background color of the entire book | Select a new **Background color**. Optionally, click **Select custom color** and use the color picker to create your own background color. |

4. For visualizations, click the visualization that you want to work with and complete one or more of the following actions.

| What to change | Actions |
|---|---|
| The color palette used in a visualization | Click the **Visualization** tab, then open the **Color** category. Click **Change color palette** and select a new palette. Optionally, click **Change heat palette** to change the heat scale order. |
| The color of the elements of a visualization (such as bars, bubbles, or lines) if the visualization does not use the **Color** data slot | Click the **Visualization** tab, then open the **Color** category. Click **Element color** and select a new color. |
| How transparent or opaque a visualization is | Click the **General** tab, then move the **Opacity** slider. |
| The fill color for a visualization | Click the **General** tab, then click **Fill color** and select a new color. Optionally, click **Select custom color** to create your own color with the **Color picker**. |
| The border color for a visualization | Click the **General** tab, then click **Border color** and select a new color. Optionally, click **Select custom color** to create your own color with the **Color picker**. |

5. For shape and text widgets that you added to the book, select the widget that you want to work with and complete one or more of the following actions in **Appearance** category.

| What to change | Actions |
|---|---|
| How transparent or opaque a shape or text is | Move the **Opacity** slider. |
| The fill color for a shape or text | Click **Fill color** and select a new color. Optionally, click **Select custom color** to create your own color with the **Color picker**. |
| The border color for a shape or text | Click **Border color** and select a new color. Optionally, click **Select custom color** to create your own color with the **Color picker**. |

## Change colors in Planning Analytics Workspace Classic

### Procedure

1. Click the **Properties** icon.

   If you don't see the **Properties** icon, Click the **Edit or preview** icon.
2. For the entire book, ensure that nothing else has focus by clicking the background, away from any visualization or object and complete one or more of the following actions in the **General** tab:

| What to change | Actions |
|---|---|
| The theme of the entire book | Click **Color and theme**, and under **Visual theme** select a different theme. |
| The background color of the entire book | Click **Color and theme**, and under **Background color** select a new color. Optionally, click **Select custom color** and use the color picker to create your own background color. |

3. For visualizations, click the visualization that you want to work with and complete one or more of the following actions:

| What to change | Actions |
|---|---|
| The color palette used in a visualization | In the **Visualization** tab, click **Color** > **Change color palette** and select a new palette. Optionally, click **Change heat palette** to change the heat scale order. |
| The color of the elements of a visualization (such as bars, bubbles, or lines) if the visualization does not use the **Color** data slot | In the **Visualizations** tab, click **Color** > click **Element color** and select a new color. |
| How transparent or opaque a visualization is | In the **General** tab, move the **Opacity** slider. |
| The fill color for a visualization | In the **General** tab, click **Fill color** and select a new color. Optionally, click **Select custom color** to create your own color with the **Color picker**. |
| The border color for a visualization | In the **General** tab, click **Border color** and select a new color. Optionally, click **Select custom color** to create your own color with the **Color picker**. |

4. For shape and text widgets that you added to the book, select the widget that you want to work with and complete one or more of the following actions in the **General** > **Appearance** tab:

| What to change | Actions |
|---|---|
| How transparent or opaque a shape or text is | Move the **Opacity** slider. |
| The fill color for a shape or text | Click **Fill color** and select a new color. Optionally, click **Select custom color** to create your own color with the **Color picker**. |
| The border color for a shape or text | Click **Border color** and select a new color. Optionally, click **Select custom color** to create your own color with the **Color picker**. |

## Enable or disable item border in consumption mode

You can enable or disable borders around items (widgets, visualizations, and explorations) with the **Show selected border in preview** property.

When the **Show selected border in preview** property is enabled for an item, a border displays around the item when it is selected in consumption mode.

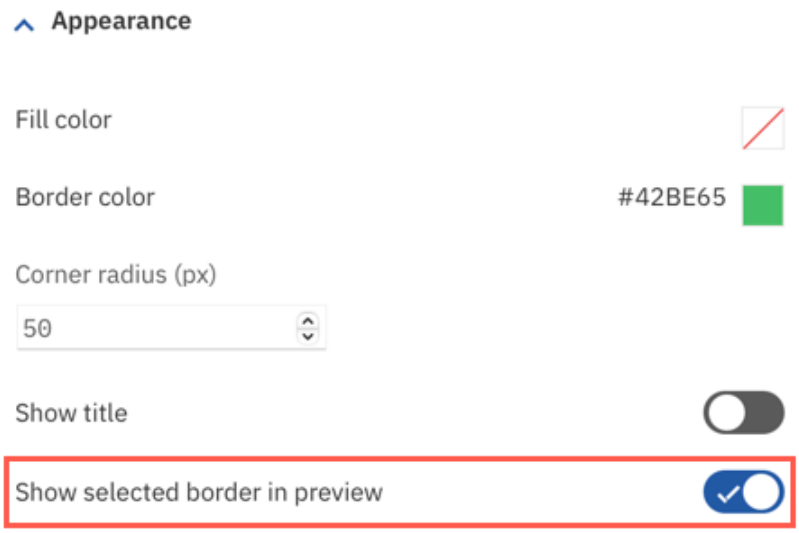
You can turn the **Show selected border in preview** property on or off for each item individually. By default, it is enabled for dynamic items like visualizations and explorations. For all other items (widgets such as Text, Image, Webpage, and Shapes) the property is disabled.

**Note:** The Line shape widget does not have a rectangle box around it when selected in **Edit** mode.

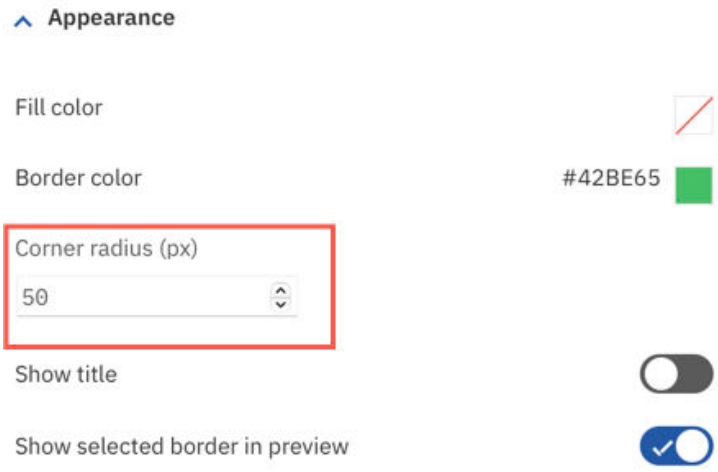
## Procedure

1. In **Edit** mode, select the item for which you want to enable or disable borders in consumption mode.
2. Click the **Properties** icon .
3. Under the **General** tab, go to **Appearance**.

   **Note:** For shape widgets, you see **Appearance** right after you click **Properties**.
4. Enable or disable **Show selected border in preview**.




5. If you want to set a corner radius for the border, enter the pixel value in the **Corner radius** field. The default corner radius value is 0 px.



# Customize tabs

Customize tab styling for individual tabs or for all the tabs in your book. You can move the tabs to the top, left, bottom, or right of your book and you can add icons to your tabs.

## Customize tab styling for individual tabs

For individual tabs on your book, you can change the tab title color, the bar under the title that indicates which tab is selected, and the tab fill color.

### Procedure

1. Open a book that has at least one tab.
2. Click the **Properties** icon.
   
3. Open the **Tabs** category.
4. Under **Individual tab styling**, click the tab that you want to customize.
5. Enter a title for the tab.
6. Select a color for the tab title.
7. Select an icon to display on the tab.
8. Select a color the icon.
9. Use **Selected bar color** to choose a color for the bar under the tab that indicates which tab is selected.
10. Select a fill color for this tab.

    **Tip:** You can also click **Select custom color** to open the color picker instead of choosing from the default color options.

## Customize tab styling for all tabs

At a global level for all tabs on your book, you can change the tab title color, the bar under the title that indicates which tab is selected, and the tab fill color.

### Procedure

1. Open a book that has at least one tab.
2. Click the **Properties** icon.
   
3. Open the **Tabs** category.
4. Under **Master tab styling**, select a color for tab titles.
5. Use **Selected bar color** to choose a color for the bar that indicates which tab is selected.
6. Select a fill color for the tabs.

   **Tip:** You can also click **Select custom color** to open the color picker instead of choosing from the default color options.

## Change tab position or order

You can move the tabs to be displayed on the top, left, bottom, or right of your book.

### Procedure

1. Open a book that has at least one tab.
2. Click the **Properties** icon.
   
3. Open the **Tabs** category.

4. From **Tab position**, select **Top**, **Left**, **Bottom**, or **Right**.
5. To change the order of tabs, click a tab and drag it to a new place within the tab order.

### Add icons to tabs

You can add icons to your tabs and control the placement and color of the icons.

#### Procedure

1. Open a book that has at least one tab.
2. Click the **Properties** icon.
3. Open the **Tabs** category.
4. Under **Individual tab styling**, click the tab to which you want to add an icon.
5. Click **Icon**, then select an icon.
6. Click **Icon color**, then pick a color for the icon.
7. Click Back to return to the **Tabs** category.
8. Use the **Icon position** option to select the position of the icon on the tab, relative to the title.

### Hiding the tabs

If your book has tabs, you might want the tabs hidden when the book is viewed by users. You can hide the tabs to create more space on the screen.

#### Procedure

1. Open a tabbed book in edit mode.
2. Click the **Properties** icon.
3. Open the **Tabs** category.
4. Click the **Show tabs in preview** toggle to hide the tabs.

## Change the template on a tabbed book

You can change the template while you're assembling a tabbed book. After you change the template, move the objects around to fit.

#### Procedure

1. While in edit mode, click a tab.
2. Click **Change template**.
3. Select a template.
4. Click **Create**.

## Enable pinning in consumption mode

You can enable pinning of items in consumption mode.

By default, pinning is available in edit mode. For more information, see "Pin items" on page 671.

#### Procedure

1. Open a book, and make sure that nothing is selected on the canvas.
2. Click **Properties**.

The **Dashboard properties** panel opens.

3. On the **General** tab, expand **Advanced**.
4. Enable the **Allow pinning in consumption** property.

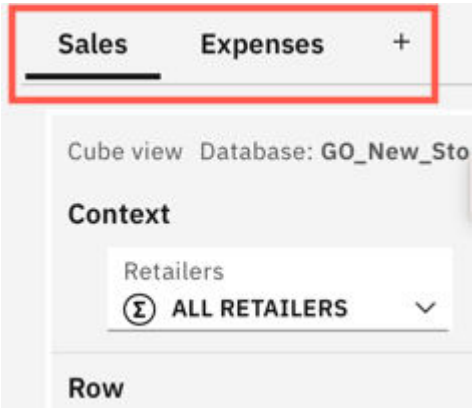
# Set book properties

You can set properties to achieve a consistent look for a book. Select the theme, background color, layout, tab styling, and the synchronization scope for a book.

## Set book properties inPlanning Analytics Workspace

### Procedure

1. In Edit mode, click the **Properties** icon.
2. In **General**, you can set the following properties:
   a) Under **Color and theme**, you can select the **Visual theme** and **Background color** for the book.
   In this section, you can also select a background color for each individual tab or set a default background color for all tabs.
   b) Under **Tabs labels**, decide whether you want to **Show tabs in preview**. To hide tabs, turn off the **Show tabs in preview** option. In the **Individual tab styling** section, you can rename tabs, select tab title and background color, and add icons next to tab titles.
   Tab titles are always visible when you are in Edit mode, so to see what the dashboard looks like, disable Edit mode. You can free up space in the book by hiding tab titles.
   **Tip:** If you hide the tab titles, add navigation buttons to your view so that you can move between tabs when the tabs aren't visible in preview. To find out more, see “Action Buttons ” on page 652.
   The following image shows a view with tab titles:




3. In **Synchronization**, select the level of synchronization by setting **Synchronization scope** to either **Book** or **Sheet**.
   To learn more, see “Synchronize objects in a book or sheet” on page 593.

# Add pictures, media, and web pages

You can add pictures, media, and web pages to your Planning Analytics Workspace books. For example, you can add a company logo, and a video blog that explains results.

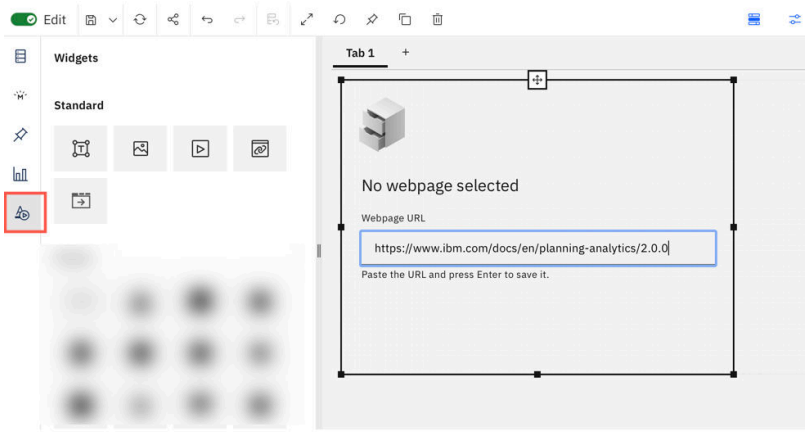
**Note:** You must have permission to use the content that you add. The content must be available from an address beginning with HTTPS, and the publisher of the content must allow it to be displayed in a frame.

There are two ways to add pictures, media, and web pages to your book: **Insert content as a widget** or **copy and paste content**.

## Insert content as a widget:

The item that you want to add must be available from a URL. In a browser window, locate the item, and copy the URL (address).

1. Open a book in Edit mode.
2. Click the **Widgets** icon on the side bar.
3. Under **Standard**, select the type of item you want to add (**Image**, **Media**, or **Webpage**).
4. Paste the copied URL into the appropriate field.




## Copy and paste content:

You can copy and paste content directly into the book. The software automatically identifies the type of content you pasted and converts it to the appropriate widget.

1. Locate the item that you want to add to the book and copy it. You can copy and paste:
   - Images from your computer, online images from your browser, and URLs for images (image files with .jpeg, .jpg, .gif, or .png extensions)

     **Important:** The image that you paste cannot be more than 200 KB in size.
   - URLs to media (such as online videos)
   - URLs to web pages
   - Text

     The text is pasted in plain text without any formatting. To format the text, select the text that you added and click **Properties**. Under **Text details**, you can select the font, font size, text color, style, and more.
2. Open a book in Edit mode.
3. Click anywhere in book and use **CTRL+V** (Windows) or **CMD+V** (Apple Mac) to paste the copied content.

   When pasting content, make sure that any open dialog box is closed and focus is not on an input field (such as in a widget or in **Properties**).

After the item is added, click the item to reposition or resize it using its handles.

You can layer items on a sheet. For example, you might want to layer text on an image. When items are layered on a sheet, select an item and click **Send to front** or **Send to back** to change the order of the layered items.

# Add cell values to the sheet as a cell view

You can add cell values to the sheet in a book to highlight a particular value in a view.

If the cell value changes in the cube view, the cell value changes on the sheet. If you edit the cell value on the sheet, it is updated in the cube view.

You can add comments to cell values on the sheet, and comments are inherited from the cell in the cube view, see "Comments" on page 610.

You can hold cell values by right-clicking the cell and selecting **Hold**, see "Data entry" on page 606.

You can apply conditional formatting to cells. Conditional formatting is initially inherited from the cell in the cube view, but then maintained separately, see "Conditional formats" on page 626.

You can change the cell view properties, and you can also make the cell value read-only.

## Procedure

1. In Edit mode, right-click the cell that you want to add and select **Add to sheet**.

   The value is added to the sheet as a cell view that you can resize, format, and change the properties.
2. Resize the cell value by clicking it so that handles appear, then drag the handles until you get the size that you want.

   You can also position the cell view on the sheet by using the drag handle.
3. To change the format of the data in the cell, right-click the cell.

   In Planning Analytics Workspace, select **Format values**.
4. To change the cell value properties, click the cell value, then click the **Properties** icon.

   You can change the following features:

   **Exploration features**

   - **Color cells by data state** - if selected, read-only cells are colored gray, and calculated cells are colored green. Clear this option to remove color.
   - **Show filters icon** - displays the filter icon in the cell view. Clicking the displayed filter icon shows the current hierarchy dimension context of the cell.
   - **Show as cell** - when checked, the cell view appears with an editable input area. When cleared, the cell view appears like the previous non-editable read only cell view.
   - **Abbreviate numbers** - abbreviates numbers. Disable to use full numbers. When you abbreviate numbers, "K" is used to represent thousands and "M" represents millions. For example, when **Abbreviate numbers** is enabled, 20,000 appears as 20K and 15,500,000 appears as 15.5M.

   **General style**
   The fill color, the border color, and the opacity of the cell view. If **Show as cell** is enabled, the input area is not changed based on these properties.

   **Synchronize**
   You can set the Synchronize sandbox and Synchronize dimensions options for the cell view. These settings initially reflect the settings in the source cube view. So if the Synchronize sandbox setting is cleared in the cube view, it is cleared for the cell view.

   To find out more, see "Synchronize objects in a book or sheet" on page 593.

   **Text properties**
   The font size, font family, and font style.
5. To make the cell value read-only so that it can't be edited, click the cell value to select it. Click **Properties**, then click **Exploration features**, and clear **Show as cell**.

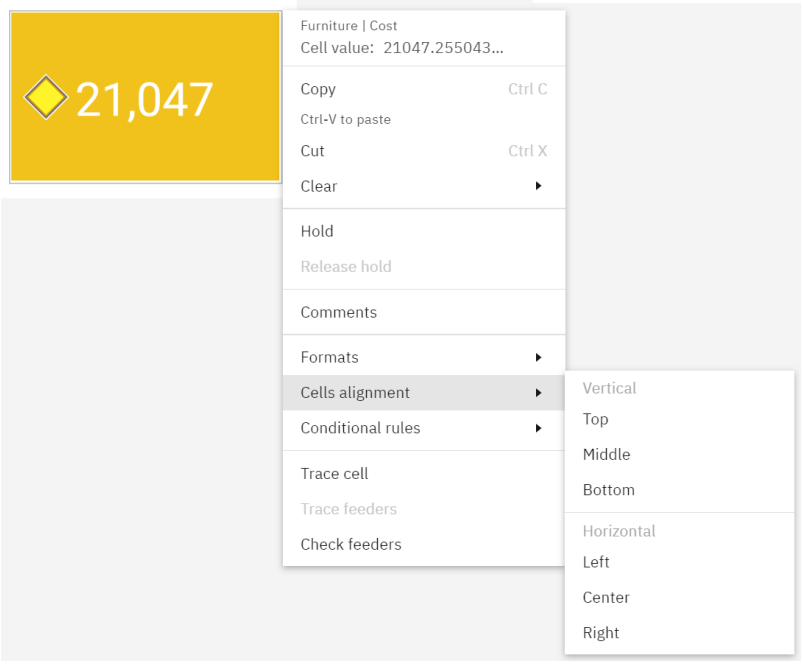
   **Note: Show as cell** changes the appearance of the cell view to look like the previously supported non-editable cell value.

### Align cells and headers in a single cell visualization

You can align the value in data cells and headers in a **Single cell** visualization either vertically or horizontally.

When you create a new single cell visualization, the value is right-aligned by default and the value remains aligned to the right as you type. This default setting maintains consistency in the alignment of values in the cube viewer and in single cell visualization.

Right-click the single cell visualization to see cell alignment options.




## Add shapes and text

You can add shapes and text to your Planning Analytics Workspace books. You can use these objects in combination with pictures and color to make engaging dashboards.

### About this task

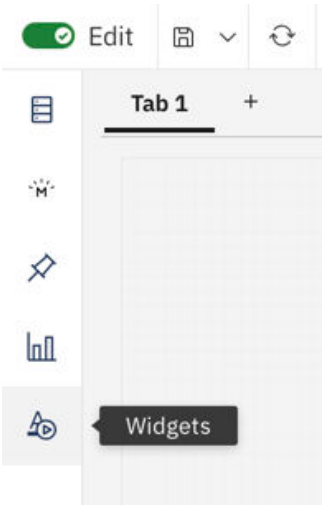
Suppose that you want to create a shape that looks like this:



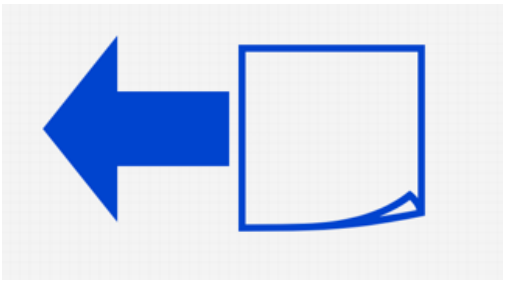

The image consists of two shapes and some text.

### Procedure

1. In Edit mode, click the **Widgets** icon.




2. Under **Shapes**, select **Left arrow** and **Sticky note 2**. The book looks like this:



To resize the widgets, select each one and use the handles to resize.

**Tip:** Use the **Send to front** or **Send to back** options in the ribbon to layer shapes with images, other shapes, text, and views.




3. To change the color of the widgets to the light blue shown in the example, select both widgets in the book and click **Properties**.
4. Under **Appearance**, select the fill color.

   **Tip:** You can also change opacity. This technique is useful if you are layering shapes over images.
5. To add text, click the **Text** icon under **Standard** in the **Widgets** panel.

   You can format the text in **Properties**.

## Adding shapes and text in IBM Planning Analytics Workspace Classic

### Procedure

1. In Edit mode, click the shapes icon on the upper right of the screen and click a shape to select it. Scroll through the shapes by clicking the arrows.

   Select this arrow shape, and this page shape. It looks something like this:



2. Press **Ctrl** and click the shapes to select them, and then click **Properties** .
3. Select the fill color, and then click **Properties** again to close it.
   The example uses light blue, with no border.

   **Tip:** You can change opacity. This technique is useful if you are layering shapes over images.
4. To make an image similar to the example, click each shape and resize them using the handles .
5. Add text by clicking **Text** .
6. Format text by clicking **Properties** with your text selected.

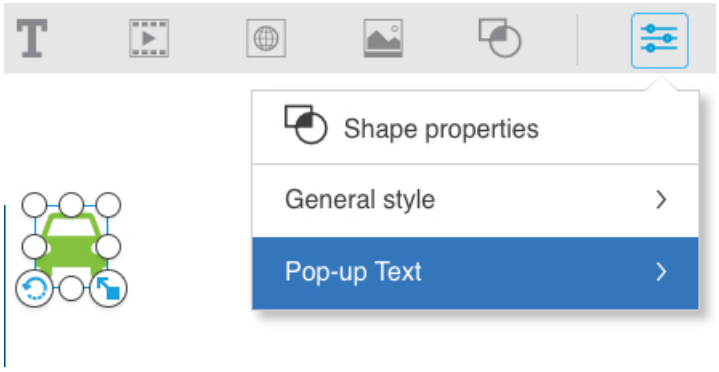
   **Tip:** You can layer shapes with images, other shapes, text, and views by changing the order of the shape in relation to the other objects. On the shortcut bar, click the object, then click the order icon .

## Add pop-up text to books

You can add explanations and guidance to books in a pop-up box. Pop-up text allows you to provide information to people about what they need to do.

### Procedure

1. In edit mode, add a shape, an image, or some text, to the sheet.
   You click the shape or text to display the pop-up box.
2. Select the shape, image, or text, and click , **Pop-up Text** > **Enable Pop-up text**.




3. Select **Preferred Position**: **Top**, **Bottom**, **Left**, or **Right**. **Preferred Position** is the position of the pop-up box in relation to the shape, image, or text.

   The Preferred Position might not be possible because of the location.

   For example, if you select **Top**, but the position of the shape, image, or text that you are clicking is at the top of the screen, the pop-up box uses the opposite position.
4. Type a title and some descriptive text for the pop-up box. You can include web addresses. Web addresses are automatically formatted as URLs, so they are clickable.
5. Click **Save**, then turn off edit mode so that you can test the pop-up box.
6. Click the shape, image, or text. The pop-up box is displayed on the screen. To move the box around the screen, or resize it, first click the pin . You can then move or resize the box. Click the pin again to move the box back to the original position next to the shape, image, or text.

# Views

A view is a defined selection of a cube and its data that can be used for analysis, exploration, and data entry. You can also have views that are used as data sources.

Views are automatically saved as part of the book that the view was created in. Views that are used as data sources are stored in the TM1 database. Find out more in “Define a cube view data source” on page 802.

To quickly add a view that can be used in a book, right-click the cube in the data tree that contains the data that you want to use, and select **Add new view**.

**Note:** When Planning Analytics Workspace opens a view, it initially fetches approximately 120 rows by 30 columns. This page size represents a compromise between performance and usability. Increasing the visible rows and columns in the view will not fetch more rows or columns from the database.

## Add a new view to a book

You can create a view or an exploration of a cube and add it to a book so that you can do ad hoc analysis.

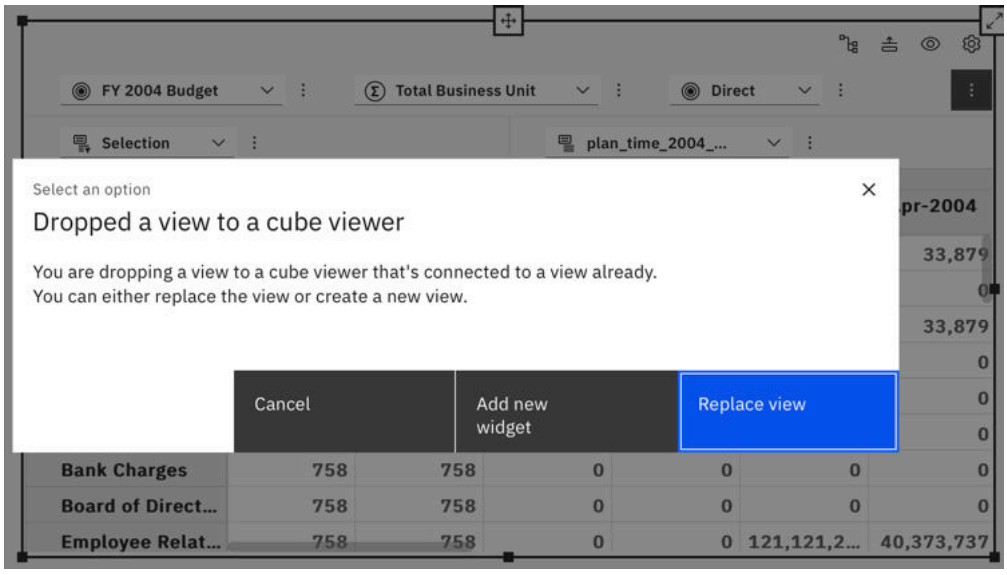
Views are automatically saved as a part of the book in which they are created. You can also save a view as a Planning Analytics Workspace object, so it can be opened from the **Welcome** page. You can share the view with other people or reuse the view in other books.

Follow these steps to add a view:

1. Open a book in **Edit** mode.
2. On the **Data** tree, right-click the cube that contains the data you want to use.
3. Click **Add new view**.

You can also add a view from **Views** node in the Data tree or drag a view and drop it on to an empty cube viewer.

If you drag a view on to a cube viewer that is not empty, you can either replace the existing view with the new one, or add the view to the book as a new widget.




**Note:** When you replace an existing view with a new view, the new view inherits the existing view’s properties (such as background color).

## Create views with Data explorer using natural language

You can now create views in books with the new **Data explorer** feature using natural language. The **Data explorer** feature uses generative AI to analyze your query and matches it to the relevant data in

TM1 to recommend views. An explanation for the logic behind the recommendation accompanies each recommended view. You can make changes to the view and add it to the book or insert it as-is.

The initial release of **Data explorer** is available in English only.

**Important:** The **Data explorer** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM® sales representative or go directly to your IBM account.

Creating views in the **Data explorer** is a two-part process:

**Analyze cube**
For **Data explorer** to recommend views, you need analyzed cubes. Cube analysis takes place in the modeling workbench, which allows AI to review the structure of a cube and understand the data.

**Submit query**
Once cube analysis is complete, you can go to a book and enter your query in the **Data explorer** to get recommended views.

## Prerequisites

Before you can use **Data explorer**, you need to have access to the analyzed cube, access to all of the dimensions and hierarchies in the cube, and read access to:

- CubeSecurity
  - }CubeAttributes
  - }DimensionAttributes
  - }ElementAttributes_<dim> (For each dimension in the cube)
  - }SubsetAttributes_<dim> (For each dimension in the cube)
- DimensionSecurity
  - }Cubes
  - }CubeAttributes
  - }Dimensions
  - }DimensionAttributes
  - }ElementAttributes_<dim> (for each dimension in the cube)
  - }SubsetAttributes_<dim> (for each dimension in the cube)
  - }Subsets_<dim> (for each dimension in the cube)

## Creating views in Data explorer

Follow these steps to run a cube analysis and then create a view in **Data explorer**:

1. In the modeling workbench data tree, go to the cube that you want to analyze.
2. From the cube's menu, click **AI operations > Analyze cube** and click **Analyze**.

   **Note:** You can check the status of a cube analysis by clicking **AI operations > Analyze log**.
3. After the cube analysis completes, go to the book where you want to create the view.
4. Click **AI operations** and select **Data explorer**.
5. Select the database and analyzed cube with the data that you want to use to create a view.

   **Note:** If you choose **Auto-select** for the analyzed cube, the most relevant cube is automatically selected to create a view, based on your query.
6. Enter your query in natural language. For example, you can enter `Show me sales performance trends over the last year` or `Show me 2024 sales of mobile devices in north america`.
7. Click **Submit**.

A view generates along with an explanation for the recommended view and the cube chosen to generate the view. You can add the view to the book in edit mode.

You can also make certain changes to the view such as switch rows and columns, move dimensions to the context area, edit the set, and change levels.

If there are other analyzed cubes with data that match your query, they are listed next to **Alternative cubes**. Click an alternative cube to see a recommended view from that cube.

### AI-generated views

The **Data explorer** feature uses granite-3-8b-instruct from the Granite family of IBM foundation models to create the view and the explanation of its recommendation.

The Granite models are decoder-only models that can efficiently predict and generate language. These models were built with trusted data that has the following characteristics:

- Sourced from quality data sets in domains such as finance (SEC Filings), law (Free Law), technology (Stack Exchange), science (arXiv, DeepMind Mathematics), literature (Project Gutenberg (PG-19)), and more.
- Compliant with rigorous IBM data clearance and governance standards.
- Scrubbed of hate, abuse, and profanity, data duplication, and blocklisted URLs, among other things.

**Note:** IBM is committed to building AI that is open, trusted, targeted, and empowering. For more information about contractual protections that are related to IBM indemnification, see the IBM Client Relationship Agreement and IBM watsonx.ai service description.

## Set properties for a view

You can change the appearance of a view, change when data is refreshed, define the synchronization scope, and allow access to the Set editor from the **Properties** menu.

### Procedure

1. In **Edit** mode, click anywhere on the view.
2. Click **Properties**.
3. Set the **Custom** properties for your view. The properties that you can set depend on the type of view; explorations (table format) have different options to other kinds of charts. .

   **Table style**
   You can show or hide row and column headers.

   You can also add border lines or zebra stripes to the table.

   **Set editor**
   Enable the **Allow access to Set editor** option to allow users to access the Set editor from the view. Disable the option to prevent access to the Set editor. When you disable access to the Set editor, users cannot open the Set editor, but they can select from members of the current set.

   **Exploration features**
   You can choose whether to display or hide the following features for explorations:

   - **Show +/- on rows**. Clearing this option removes the +/- indicator from row headers. This gives a more streamlined appearance, but you can't see if a member can be collapsed or expanded.
   - **Show +/- on columns**. Clearing this option removes the +/- indicator from column headers.
   - **Indent row levels**. Clear to remove indents from the row headers.
   - **Indent column levels**. Clear to remove indents from the column.
   - **Color cells by data state**. If selected, read-only cells are colored gray, and calculated cells are colored green. Clear this option to remove color.
   - **Bold consolidation**. When enabled, consolidated values in the view display in bold.

- **Allow consolidation type in**. When enabled, you cannot type data into a consolidated cell. This option is set as a property of an individual exploration; if a book contains multiple explorations, the property can be set independently for each exploration.

**Responsive grid**
**Responsive grid** makes the view responsive to size changes. Enable the property to set a fixed number of columns and rows and the view displays the fixed number of columns and rows when you resize the grid.

**Toolbar**

To manage which icons are available on the toolbar for you and your users, you can apply the **Toolbar** icon settings in the **Properties** panel. Click the **Custom** properties tab to view all the available **Toolbar** icons that you can set.

For a cube viewer, you can choose to display or hide the following toolbar icons:

- **Export**
- **Snap commands**
- **Save**
- **Format values**
- **Edit MDX**
- **Sandbox selection**
- **Change visualization**
- **Show/hide overview**
- **Forecast**

For visualizations, you can manage the **Save**, **Format values**, **Sandbox selection**, and **Change visualization** toolbar icons.

For all other visualizations such as selector widgets, only the **Change visualization** toolbar icon is available to toggle.

When you also select both a cube viewer and other visualizations in a book only the common toolbar icons are available.

# Save a view

You can save a view as a Planning Analytics Workspace object or as a view in a Planning Analytics database.

## About this task

When you save a view as a separate Planning Analytics Workspace object, you can share the view with other people, and reuse the view in other Planning Analytics Workspace books.

You can also choose to save a view to the Planning Analytics database. When a view is saved to the database, it can be used by processes as a data source from which you can extract data and create or update objects or data.

## Procedure

1. Click anywhere on the view to select it.
2. Click the **Save** icon on the view toolbar.
3. Choose the **Save** option.

**Save**
If the view was previously saved as a Planning Analytics Workspace object, use this option to update the view definition.

**Save view as**

If the view was not previously saved, use this option to save it as a Planning Analytics Workspace object. If the view was previously saved, use this option to save it as a new object.

Enter a **Name** for the view and choose the location where you want to save the view, either the **Shared** folder or your **Personal** folder.

**Save to database**

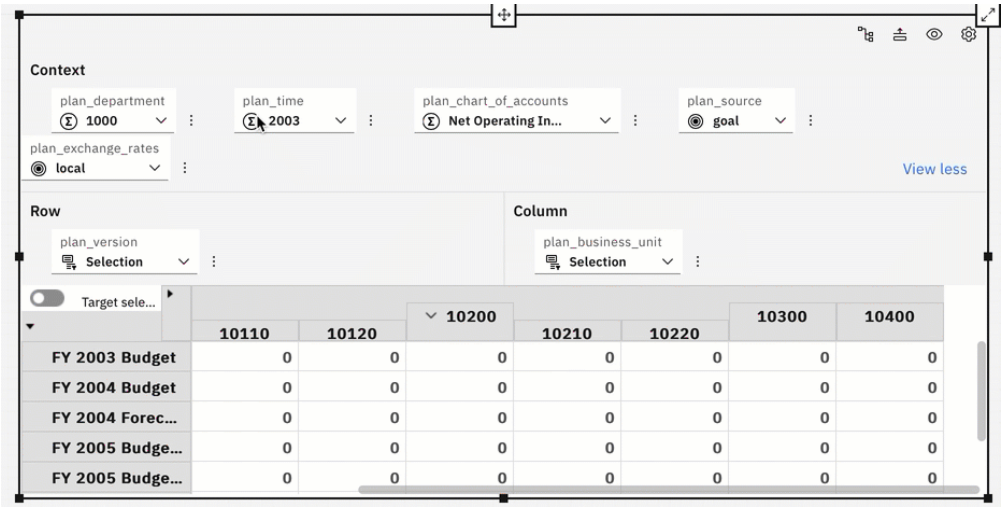
Use this option to save the view to the Planning Analytics database. Views saved to the database can be used only in Planning Analytics for Excel, Planning Analytics Workspace, and as a process datasource.

- Select the **Private** option if you want to save the view as a private view that is available only to yourself. If the **Private** option is not selected, the view is saved as **Public** and it is available to any user who has at least read access to the cube that contains the view.
- Select the **Native** option if you want to save the view as a TM1 native view. If the **Native** option is not selected, the view is saved as **MDX**.

## Change the display of data in a view

Dimensions can be moved to different positions in a view.

You can change the position of a dimension in a view by dragging the dimension tile and dropping it into a new position in the row, column, or context area. When you drag a dimension from the row or column area to the context area, the view focuses on that member.




Dimensions that are used as context can be visible in the view or can be placed on the bench to save space and to simplify the appearance of the view.



**Note:** If you move a dimension from a row or column to the context area, any filters, sorts, or keep, show or hide actions that have been applied to the dimension using the right click menu from the exploration view, are removed.

You can also use the Fields panel to manage settings for a view. All of the actions that are available in the overview area of a view are supported in the **Fields** panel, except **Move to bench**.

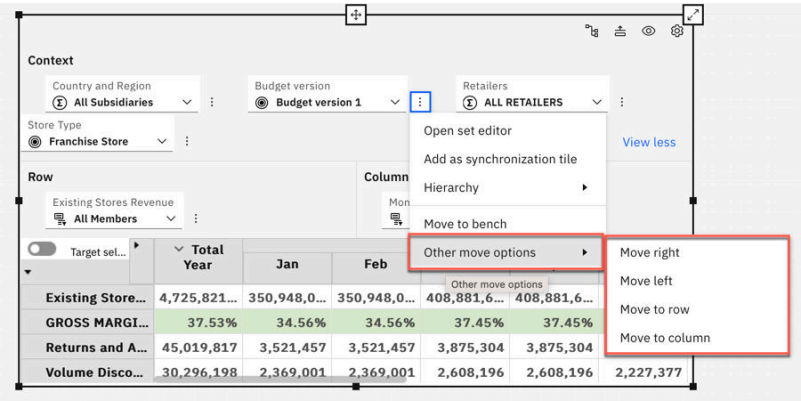
### Procedure

1. Create a book.
2. Navigate to the **GO_New_stores** database, **Base Sales Forecast** cube in the tree, and add the **All** view to the book.

   This view has one dimension on the row axis, one dimension on the column axis, and four dimensions as context.
3. Click and hold the **Retailers** dimension in the context area, then drop it next to the **Month** column dimension.

   You can drag a dimension from any location in a view to a different location. When you drop one dimension directly on top of another dimension, the positions of the dimensions in the view are switched. When you drop one dimension next to another dimension, both dimensions appear in the same location in the view.
4. Click the menu icon next to a dimension to see **Other move options**.

   Depending on its position, you can move the dimension left or right, and to a row, column, or context.




## Selecting a member or set in the cube viewer

The cube viewer offers a quick way to select a member or set. This can easily be done by typing a member name or ID or set name in an exploration view.

You can open an input box from a dimension tile in the **Context** or **Bench** area to enter a member name or Member ID and update the view with that member. Similarly, you can open the input box from a tile in the **Row** or **Column** area and type the set name to update the view with that set.

This feature is especially useful if you are familiar with the data in the cube and want to avoid using the **Search** feature to find and select a member or set.

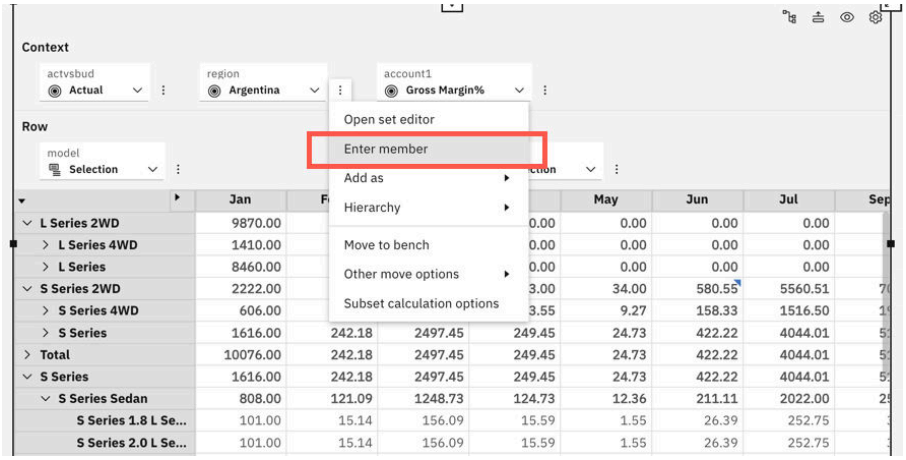
When you enter a member using the input box, the current set is bypassed and you can directly use any member. The set then changes to a single member set.

If you are selecting a set using the input box, the named set you enter must exist. If customized sets are defined, then the named set must be in the customized set list.

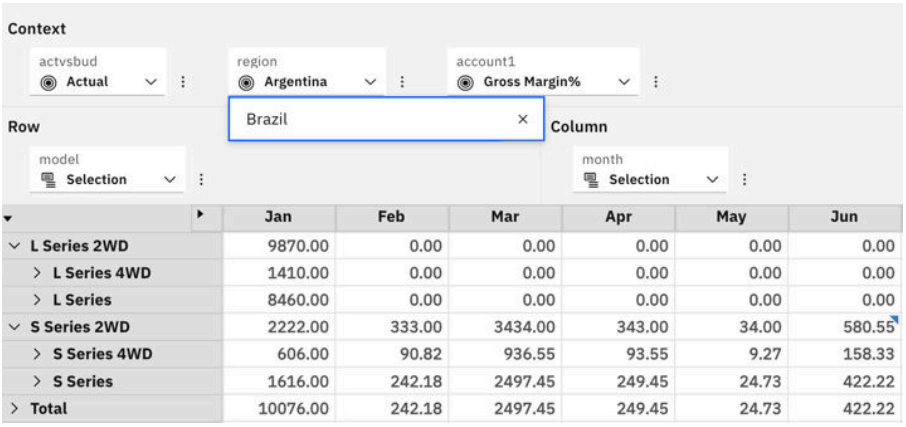
To use the input box to select a member:

1. Click the menu icon next to the dimension tile in the **Context** or **Bench** area and select **Enter member** to open the input box.

   **Tip:** If you're on Windows, simply press **Ctrl** and click the member name in a dimension tile to open the input box.

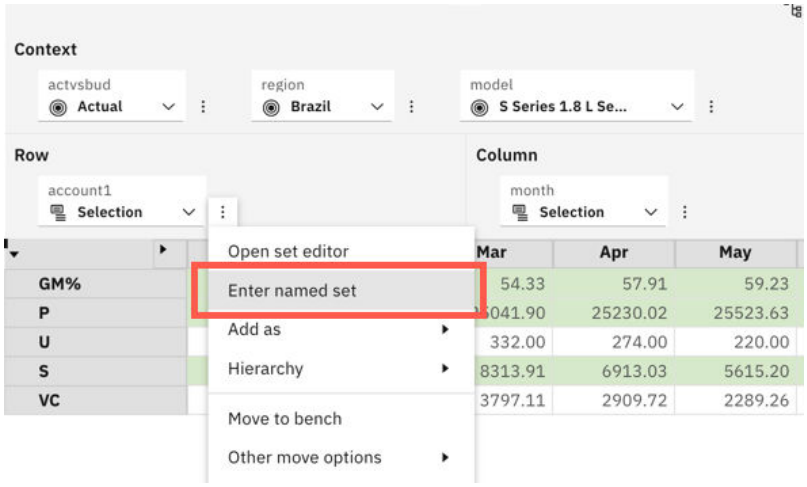



2. Type the member name or ID in the input box and press **Enter**.



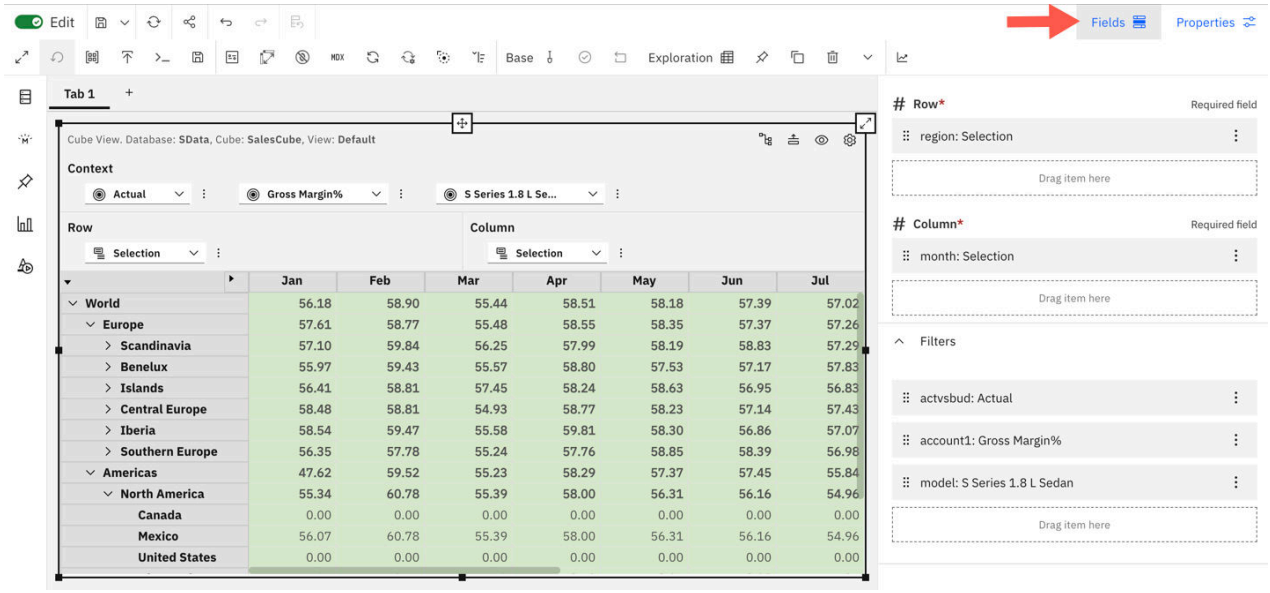

The view updates with the member you entered.

You can select a set by typing its name in the same way from the tiles in the **Row** or **Column** area. If you are using the menu option, select **Enter named set** instead to open the input box.



## Use Fields panel to change the display of data in a view

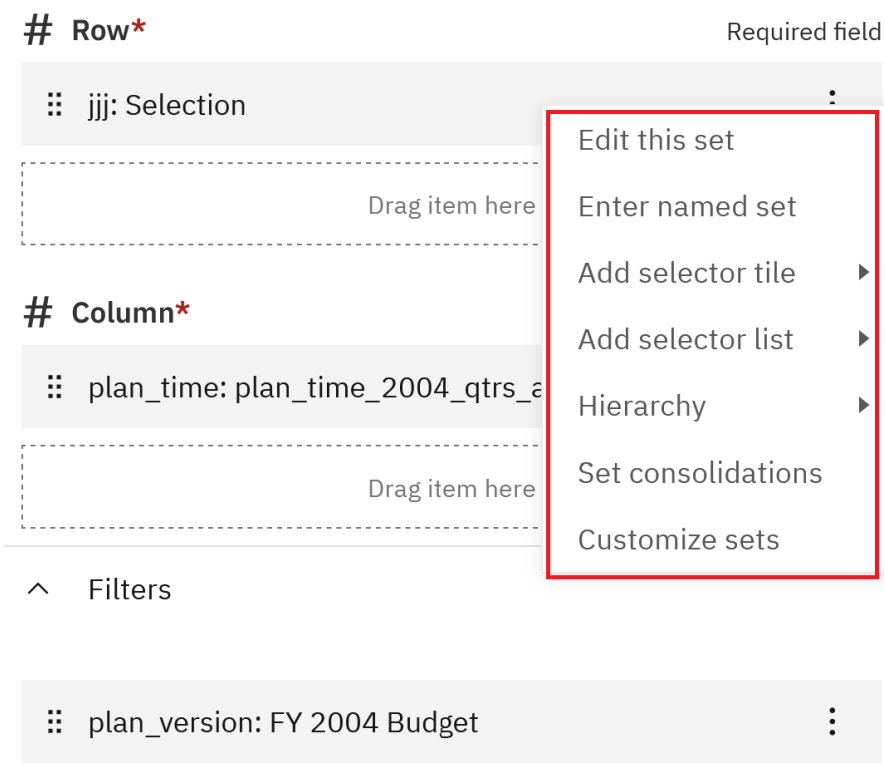
You can also change the display of data in a view with the **Fields** panel in books.




All of the actions that are available in the Overview area of an exploration are supported in the **Fields** panel, except **Move to bench**.

Select an exploration and go to **Fields** to open the fields settings. Dimensions that are in the **Context** area of the exploration appear under **Filters** in the **Fields** panel. You can drag dimensions from **Filters** to the **Row** and **Column** fields.

Click the **More** icon next to a dimension in the **Row** or **Column** field to edit the set, remove the hierarchy, create a subset calculation (set consolidations), and customize sets. You can also add a Selector list or Selector tile to the book from this menu.



## Create a Selector visualization to manage views

A **Selector** is a unique type of visualization that lets you pick members, sets, or levels for views (either exploration or visualization) in a book. When a selector is synchronized with a book or a sheet, a selection in the selector is applied to the views in the book or sheet. For example, suppose that you have an exploration or visualization and the overview is hidden. You can add a selector to the book, then click an item in the selector to change the data in the view.

### About this task

All selectors can be customized with the options available on the **Custom** tab of the visualization properties pane.

1. While a book is in edit mode, click the selector visualization,
2. Click **Properties**.
3. Click the **Custom** tab.

There are several property categories available on the **Custom** tab. You might need to expand a category to reveal all available options.

**Selector style**

**View as selector list** - Enable this property to display a list of items (sets, levels, members) on the selector. Disable to display a single item on the selector, which you can click to expand and reveal additional items.

**Show header** - Enable this property to display a header that indicates the parent dimension and hierarchy for the selector. Disable to hide the parent dimension and hierarchy.

**Show icon** - Enable this property to display an icon with each item in the selector. Disable to display only item names.

**Show overflow menu** - Enable this property to display the overflow menu at the top of the selector.

The overflow menu appears as three vertically stacked dots .

When the he **Selection type** is set to **Members**, the overflow menu provides a single option, **Open set editor**. You can open the set editor and modify the set that is displayed in the selector.

When the he **Selection type** is set to **Sets**, **Levels**, or **Sets and levels**, the overflow menu provides two options: **Open set editor** and **Customize sets**. Click **Customize sets** to modify the list of sets or levels that is displayed on the selector.

**Tip:** If the **Allow access to set editor** property is disabled, the **Show overflow menu** is automatically disabled.

**Show search bar** - Enable this property to display a search bar, which searches current items on the selector. Disable to hide the search bar.

**Allow expand/collapse** - This property is available only when the **Selection type** is set to **Members**. Enable this property to allow expand or collapse when you click a consolidated member on the selector. Disable to prevent expand or collapse on a consolidated member.

**Show indent** - This property is available only when the **Selection type** is set to **Members**. Enable this property to show indentation at each consolidation level. Disable to show members as a flat list without indentation.

**Responsive list**

**Responsive list** - This property is available only when the **View as selector list** property is turned on.

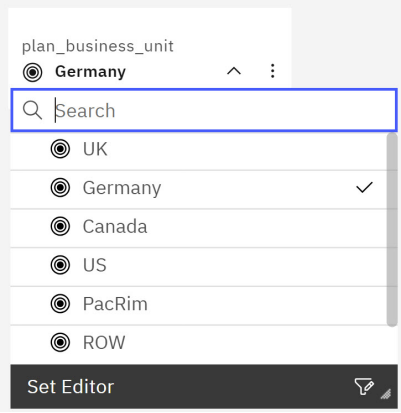
Enable this property to make the selector responsive to size changes. When this property is enabled, the selector always shows the number of items specified by the **Maximum number of items** property. If the selector is re-sized, the text size for each item is scaled to occupy all available space in the list.

Disable to maintain constant text size for items in the selector list.

**Maximum number of items** - This property is available only when the **View as selector list** property is turned on and the **Responsive list** property is enabled. Set this property to the number of items you want to see in the selector list.

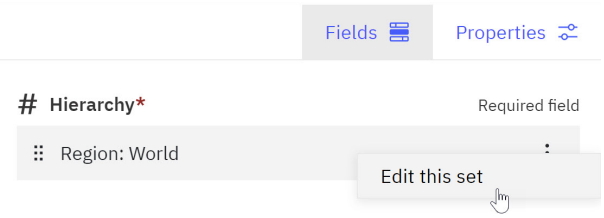
**Set editor**

**Allow access to set editor** - Enable this option to allow access to the set editor both from the overflow menu and from the list of members on a selector that is formatted to display a single item with a drop-down menu.




Disable this property to prevent access to the set editor from the selector. When this property is disabled, the **Show overflow menu** property is automatically disabled.

If you disable the **Show overflow menu** property, but enable the **Allow access to set editor** property, the set editor is available only on a selector that is formatted to display a single item with a drop-down menu. In this case, the set editor is also available from the **Fields** panel.




**Selection type**

**Sets** - Show sets in the selector.

**Levels** - Show levels in the selector.

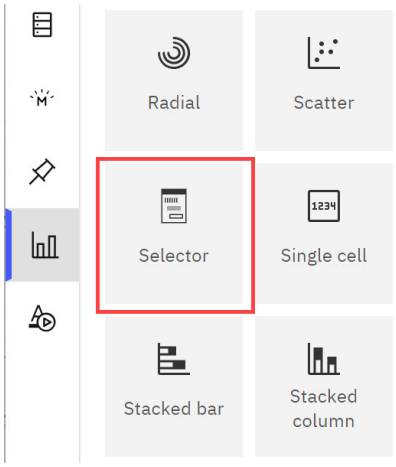
**Sets and levels** - Show sets and levels in the selector.

**Members** - Show members in the selector.

## *Create a Selector visualization from the Visualizations pane*

## Procedure

1. Click the **Visualizations** icon on the sidebar to open the pane.
2. Click **Selector**.




A selector placeholder is created on the book.

3. From the **Data** tree, drag and drop a dimension, hierarchy, set, or level onto the visualization placeholder.

### *Create a Selector visualization from the data tree*

## Procedure

1. On the **Data** tree, fully expand the **Dimensions** node beneath the cube for which you want to create a selector visualization.

   You can create a selector only from the Dimensions node beneath a cube. You can't create a selector from the main Dimensions node.
2. Right-click the dimension, hierarchy, or subset that you want to use as a selector.
3. Click **Add selector tile** or **Add selector list** and select the tile or list type. You can add the following types of selectors to your book:

   a) single tile

   b) selector with multiple tiles

   c) selector with range selection tile

   d) single list

   e) selector with multiple lists

   f) selector with multiple selection option

   **Note:** The range selection tile and multiple selection list selector types can only send synchronization events. They don't receive synchronization events.

### *Create a Selector visualization from an exploration*

## Before you begin

There are two ways to create a Selector visualization from an exploration. You can create a Selector by simply clicking the dimension tile that you want to use as a selector, then dragging the dimension to an empty space on the book. When you create a Selector from a context dimension tile, the Selector contains members. When you create a Selector from a row or column dimension, the Selector contains levels and sets.

You can also create a Selector from the menu of a specific dimension in the exploration.

## Procedure

1. In an exploration, click the menu icon next to the dimension that you want to use as a selector.

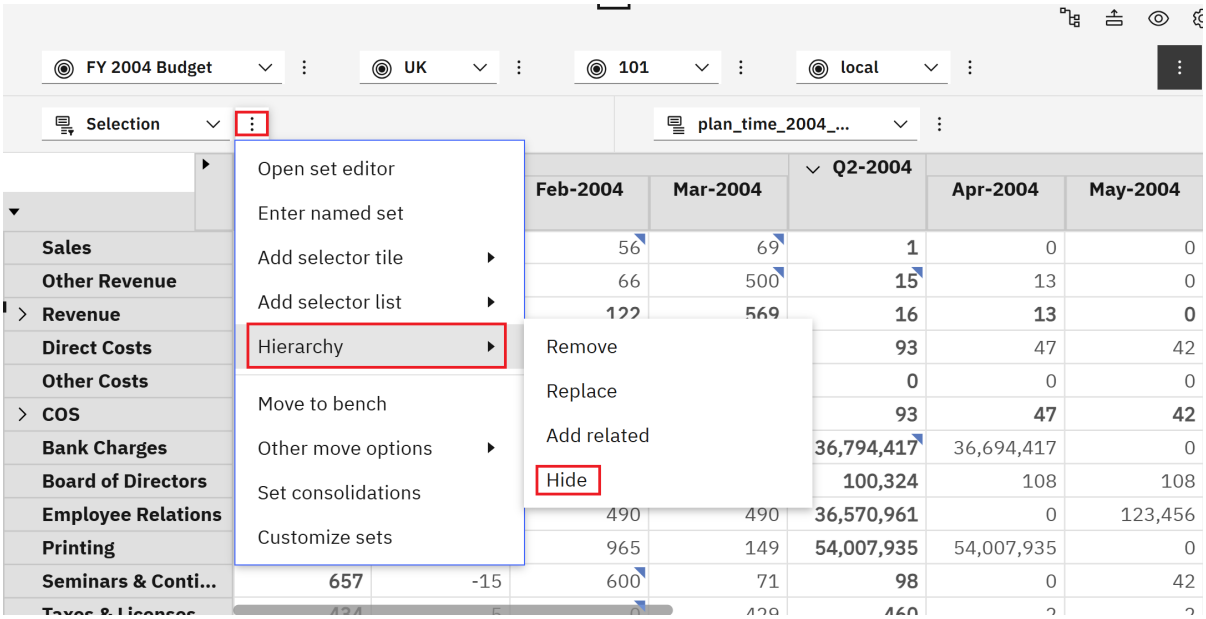
2. Click **Add selector tile** or **Add selector list** and select the tile or list type. You can add the following types of selectors to your book:
   a) single tile
   b) selector with multiple tiles
   c) selector with range selection tile
   d) single list
   e) selector with multiple lists
   f) selector with multiple selection option

   **Note:** The range selection tile and multiple selection list selector types can only send synchronization events. They don't receive synchronization events.

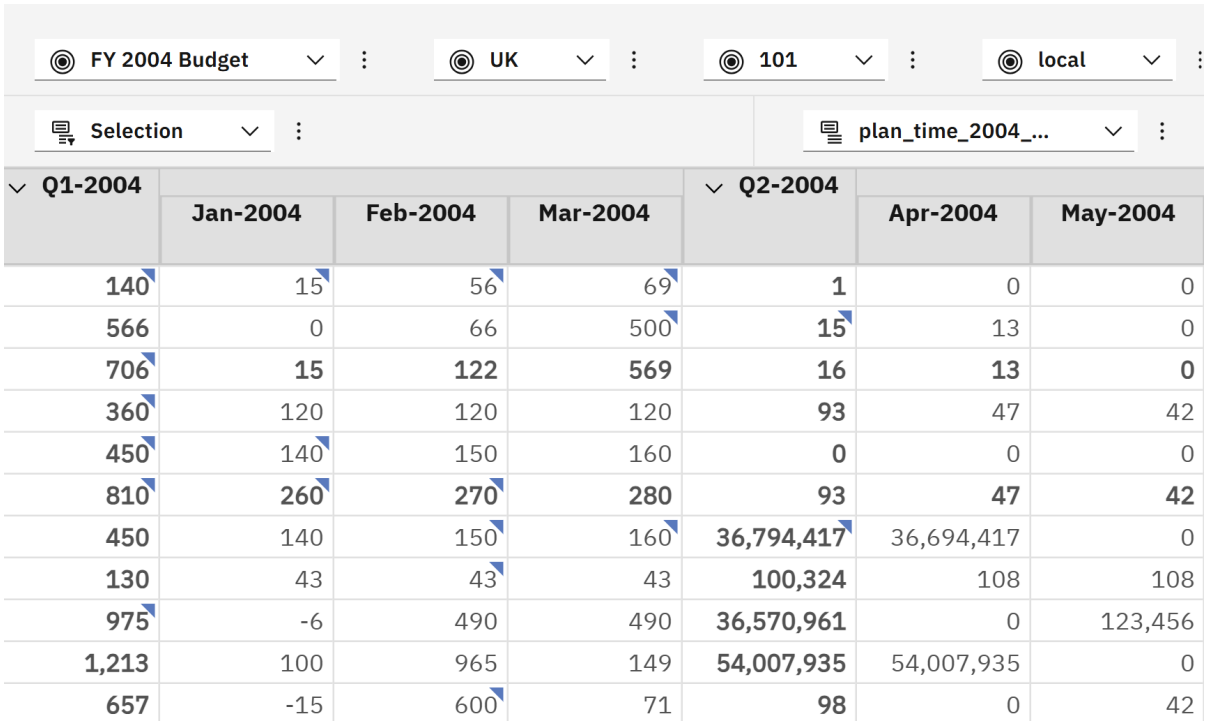
## Hide row and column headers in a view

Users can hide row and column headers in a view without triggering a data update.

To hide a row or header, from the dimension tile context menu, click **Hierarchy** > **Hide**.



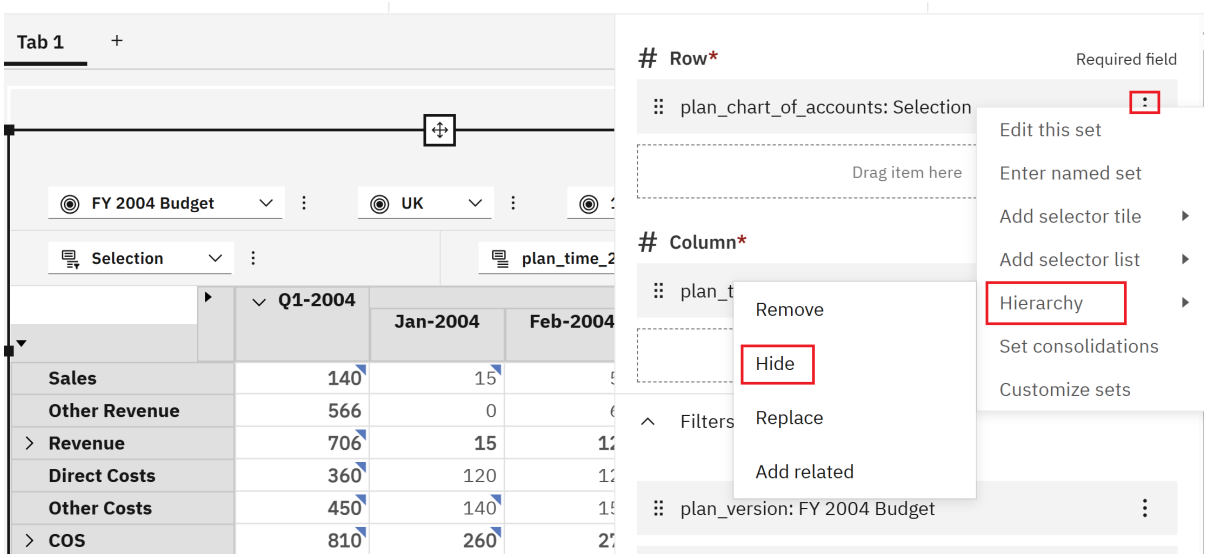

In the following image, the row headers are hidden.



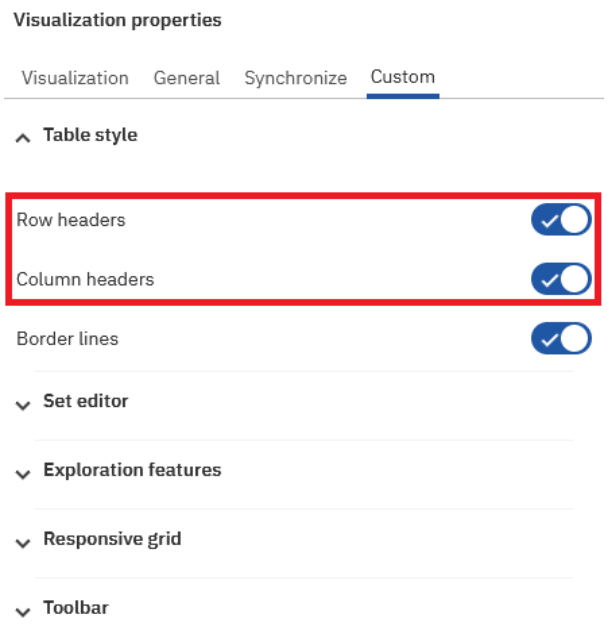

| Q1-2004 | Jan-2004 | Feb-2004 | Mar-2004 | Q2-2004 | Apr-2004 | May-2004 |
|---|---|---|---|---|---|---|
| 140 | 15 | 56 | 69 | 1 | 0 | 0 |
| 566 | 0 | 66 | 500 | 15 | 13 | 0 |
| 706 | 15 | 122 | 569 | 16 | 13 | 0 |
| 360 | 120 | 120 | 120 | 93 | 47 | 42 |
| 450 | 140 | 150 | 160 | 0 | 0 | 0 |
| 810 | 260 | 270 | 280 | 93 | 47 | 42 |
| 450 | 140 | 150 | 160 | 36,794,417 | 36,694,417 | 0 |
| 130 | 43 | 43 | 43 | 100,324 | 108 | 108 |
| 975 | -6 | 490 | 490 | 36,570,961 | 0 | 123,456 |
| 1,213 | 100 | 965 | 149 | 54,007,935 | 54,007,935 | 0 |
| 657 | -15 | 600 | 71 | 98 | 0 | 42 |

To show the row or column header, from the dimension tile context menu, click **Hierarchy** > **Show**.

The same options are also available from the **Fields** panel.




The **Hide** and **Show** context menu options are available only when the **Row headers** and **Column headers** visualization properties are turned on the **Custom** tab, as shown in the following image.




# Setting view options

You can individually enable and disable view options that manage the appearance of the context, row, and column sections of the cube viewer.

## Procedure

1. Click **View options** .
2. Enable or disable view options by clicking the check box that corresponds to the view option you want to toggle.

   **Hierarchy names**
   When you enable **Hierarchy names**, the name of the parent hierarchy appears above each member or selection name.

   **Dimension icons**
   When you enable **Dimension icons**, an icon indicating the type of member or selection appears with the member or selection name.

   **Context information**
   When you enable **Context information**, the name of the view, which database it is in, and the parent cube name are displayed.

   **Section headers**
   When you enable **Section headers**, the context, row, and column headers appear for the corresponding sections.

   **Section icons**
   When you enable **Section icons**, icons representing the context, row, and column sections are displayed.

# Change the format of data in a view

You can set the format of data in a view, overriding existing formatting in a dimension. A number of built-in formats are available, or you can specify a custom format.

**Note:** Data in a view is displayed in accordance with your browser locale setting that uses the decimal and thousands separators for your locale. You can double-click a cell to view the raw data value as it exists on the Planning Analytics database, which uses a period as the decimal separator and omits a thousands

separator. The raw value might include more decimal places than are displayed in the cell. When you enter data directly in a cell, you must use the decimal separator for your browser locale.

When you set the format in the view, the raw data format or cell data type is not changed. You can reset the format in the view to the underlying raw data format by selecting **Use database format**.

**Note:** If you're using the Format manager in Planning Analytics Workspace 2.0.94, you can reset a format in the view by restoring it to default.

Formats that are saved to a view in IBM Planning Analytics Workspace are not available if the view is opened in IBM Planning Analytics for Microsoft Excel.

Generally, when more than one format is set in a view, the format that is applied to a cell is determined in the following order of precedence:

1. The column member inner to outer axis (column dimension in the overview, from right to left).
2. The row member inner to outer axis (row dimension in the overview, from right to left).
3. Formatting that is applied to the entire view from the format button.

   In Planning Analytics Workspace, it's the **Format manager** button  on the application toolbar.
4. Context dimensions in the overview, from right to left.
5. Bench dimensions from bottom to top.

There are some unique format interactions in the case of column or row formatting versus entire view formatting. When you apply view formatting, it overrides all column or row formatting. However, any column or row formatting that is applied **after** the view formatting is applied will take precedence, in accordance with the order of precedence previously described.

You can have two copies of the same view and apply different formatting to each view. For example, you have a view of a bank in Canada that shows *Total Current Assets* in thousands. You can make a duplicate of this view, and change it to show worldwide bank data. When you duplicated the view, the formatting was the same in the copy. But you can change the formatting in the copy to show the *Total Current Assets* in millions.

The built-in formats are:

**General**
Displays numbers without commas to separate digits to the left of the decimal point. Negative values are prefixed with a minus sign (-).

```
-1234.57
```

**Fixed**
Displays numbers without commas to separate digits to the left of the decimal point. Negative values are surrounded by parentheses.

```
(1234.57)
```

**Comma**
Commas separate every third digit to the left of the decimal point.

```
(1,234.57)
```

**Rounded**
Commas separate every third digit to the left of the decimal point, rounded up to the nearest whole number. You can see the actual value by right-clicking in the cell.

```
(12,346)
```

**Percentage**
Multiplies numbers by 100 and displays a following percent sign (%). Digits to the left of the decimal point do not use commas, and negative values are prefixed with a minus sign (-).

```
-123456.70%
```

**Scientific**
Displays numbers in scientific notation. Scientific notation is a way of expressing large or small numbers. For example, the number 123,000,000,000 can be written as 1.23E+11.

Negative values are prefixed with a minus sign (-).

`-1.2E+3`

**Accounting**
Displays numbers with currency symbols and decimal points in a column. Negative values are surrounded by parentheses.

`$(1,234.57)`

**Currency**
Displays numbers with the currency symbol that is specified for your computer. Commas separate every third digit to the left of the decimal point. Negative values are surrounded by parentheses.

`($1,234.57)`

**Currency (rounded)**
Displays numbers with the currency symbol that is specified for your computer. Commas separate every third digit to the left of the decimal point, rounded up to the nearest whole number. You can see the actual value by right-clicking in the cell. Negative values are surrounded by parentheses.

`($1,235)`

**Date**
Gives you a date picker, and displays dates in a predefined format: `mm/dd/yyyy`.

`01/23/1989`

**Time**
Displays time in a predefined format: `hh:mm:ss`.

`12:30:00`

**Thousands**
Displays thousands as K.

`12K`

**Millions**
Displays millions as M.

`12M`

**Short abbreviations**
Depending on how large the number is, it's shortened by using a locale-specific abbreviation. In English, it's K for thousands, M for millions, B for billions, and T for trillions. The following table shows examples of abbreviated values in English.

| Value | Abbreviated value |
|---|---|
| 150 | 150 |
| 1500 | 1.5K |
| 1500000 | 1.5M |
| 1500000000 | 1.5B |
| 1500000000000 | 1.5T |
| 1500000000000000 | 1,500T |

Other locales don't follow the same pattern.

**Note:** The **Thousands** and **Millions** options use the K and M abbreviations for all locales.

You can also format string headers and data cells in a view to wrap text. You can wrap multi-line text in data cells with or without line breaks.

**Note:** The wrap text feature in data cells is only supported for text that is entered using the text editor (press F2 or click the pencil icon in the cell).

## Custom data formats in views

You can change the way that data is formatted in a view by creating a custom format, if the data format that you want isn't available in the built-in formats.

Creating a custom format can be useful when working with views. For example, you might want to specify a particular currency symbol, such as the Euro (€), or use a different date or time format.

**Important:** When you set the format in the view, the underlying TM1 format or TM1 cell data type is not changed. You can reset the format in the view to the underlying TM1 format by selecting **Use database format**.

Custom formatting in IBM Planning Analytics Workspace views uses ICU syntax. For detailed information, go to the following website: https://unicode-org.github.io/icu/userguide/strings/unicodeset.html.

Custom formats use a specific pattern, and many characters in a pattern are taken literally, so they appear unchanged. The following example shows how you can insert the word Total into rows or columns, and it also shows how to insert a currency symbol (€) into the value.

| **Example custom format** | **Result** |
|---|---|
| Total: €#,##0;(€#,###) | Value: `28.50`<br>Displays: `Total: €29` |

Notice that the # and 0 characters are used to represent digits. The # and 0 characters are examples of special pattern characters. The following table provides a summary of the special characters that are used in custom formats.

*Table 3. Special pattern characters*

| **Symbol** | **Meaning** |
|---|---|
| `0` | Digit<br>Can be used in combination with #. For example, `#,##0.00;(#,##0.00)`<br>Displays insignificant zeros if a number has fewer digits than there are zeros in the format string.<br>If a number has more digits to the right of the decimal point than there are placeholders in the format string, the number rounds to as many decimal places as there are placeholders. If there are more digits to the left of the decimal point than there are placeholders, the extra digits are displayed.<br>Examples:<br>Custom format: 0.00<br>Value: 23.896<br>Displays: 23.90 |

| *Table 3. Special pattern characters (continued)* | |
|---|---|
| **Symbol** | **Meaning** |
| # | Digit, zero shows as absent<br>Can be used in combination with 0.<br># displays only significant digits. In the value .90, the 0 is considered insignificant. The value is displayed as .9 when # placeholder is used.<br>If a number has more digits to the right of the decimal point than there are placeholders in the format string, the number rounds to as many decimal places as there are placeholders. If there are more digits to the left of the decimal point than there are placeholders, the extra digits are displayed.<br>Example:<br>Custom format: #.##<br>Value: 123.896<br>Displays: 123.9 |
| 1-9 | '1' through '9' indicate rounding |
| @ | Significant digit<br>Example:<br>Custom format: @@@<br>Value: 123.896<br>Displays: 124 |
| . | Decimal separator or monetary decimal separator |
| - | Minus sign |
| , | Grouping separator<br>Example:<br>Custom format: #,##,##0<br>Value: 123456789<br>Displays: 12,34,56,789<br>You can also use trailing commas at the end of a numeric format pattern to display values in compact format, For more information, see Custom compact numeric format. |
| E | Separates the mantissa and exponent in scientific notation |
| + | Prefix positive exponents with the localized plus sign |

| *Table 3. Special pattern characters (continued)* | |
|---|---|
| **Symbol** | **Meaning** |
| ; | Separates positive and negative sub-patterns<br>If you don't specify how negative numbers are shown, positive formatting is used, prefixed by the minus sign.<br>Example:<br>Custom format: #,##0.00;(#,##0.00)<br>Positive numbers: 123,456,789.00<br>Negative numbers: (123,456,789.00) |
| % | Multiply by 100 and show as percentage |
| ' | Use single quotation marks to include special characters in a prefix or suffix<br>Example:<br>Custom format: "'$'#"<br>Value: 123<br>Displays: $123.<br>To display a single quotation mark itself, use two single quotation marks in a row. For example, for the value 7, the pattern # o''clock displays 7 o'clock. |

## Custom compact numeric format

You can append one or more commas to the end of a custom number format pattern to display values in compact format in explorations and visualizations. Each comma present at the end of a format pattern scales the value display by 1,000.

A single comma results in the value being displayed as thousands. Two commas display the value as millions. Three commas display the value as billions.

For example, the format pattern #,##0.0, results in the value 1234567.89 being displayed as 1,234.6. The format pattern #,##0.0,, displays the same value as 1.2.

To improve the usability of compact format, you can include literals in the format pattern to display specified characters. You can use these characters to indicate that the value is displayed as thousands, millions, or billions. Literals must be enclosed in single quotation marks.

To improve upon the previous example, use the format pattern #,##0.0,'K' to display the value 1234567.89 as 1,234.6K. The format pattern #,##0.0,,'M' displays the same value as 1.2M.

## Date and time examples

For custom date formats, you use a date pattern. In a date pattern, strings of characters are replaced with date and time data.

| **Example** | **Result** |
|---|---|
| hh:mm a | • 12:00 AM<br>• 06:00 PM |
| EEE, MMM d, ''yy | Mon, Oct 30, '15 |

For more information, go the https://unicode-org.github.io/icu/userguide/format_parse/datetime website, and look for *Date/Time Format Syntax*.

## Setting the data format in a view

You can set the data format on a row or column, or on the entire view.

Data formatting can be done in the view (exploration) or visualization. The formats are retained when the view is switched back to exploration or other type of visualization.

### Procedure

1. Do the following steps to set format on a row or column in the exploration or visualization:
   a) Right-click the column or row header, and click **Format values**. If you are using the new experience view, click **Formats** instead.

      For visualizations, you can access the **Format values** menu from the data item context menu ⋮ under the **Fields** tab.
   b) Select the format, such as **Currency**.

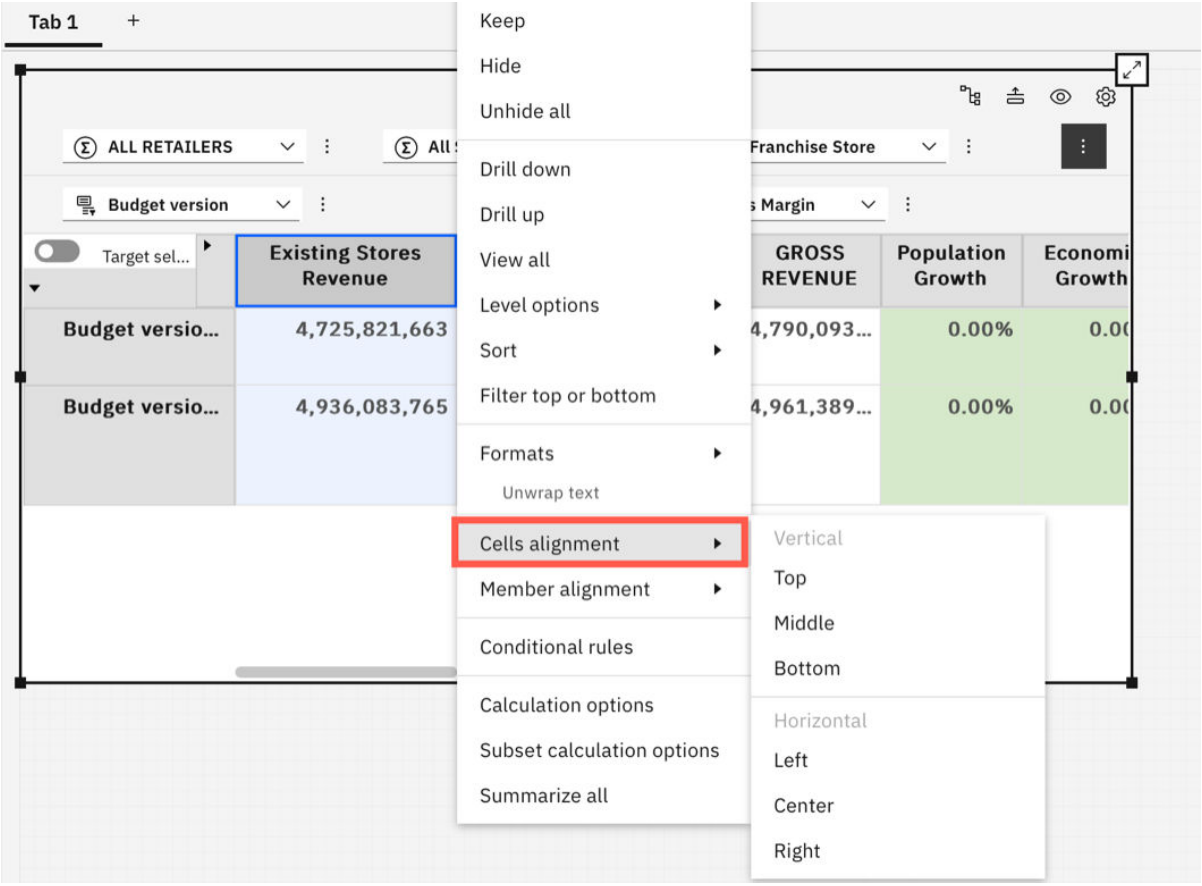
      You can apply formatting to both columns and rows. Column formatting takes precedence over row formatting. For visualizations, you can also set the data format for the axes.
   c) If none of the built-in formats are suitable, select **Custom format** and type the pattern for the custom format.
   d) To reset the format that is used to the database default, select **Use database format**.
2. To set format on the entire view:
   a) Click the handle, then click the **Format values** icon in the application toolbar.
   b) Select the format, such as **Currency**.
   c) If none of the built-in formats are suitable, select **Custom format**, and type the pattern for the custom format.
   d) To reset the format that is used to the database default, select **Use database format**.

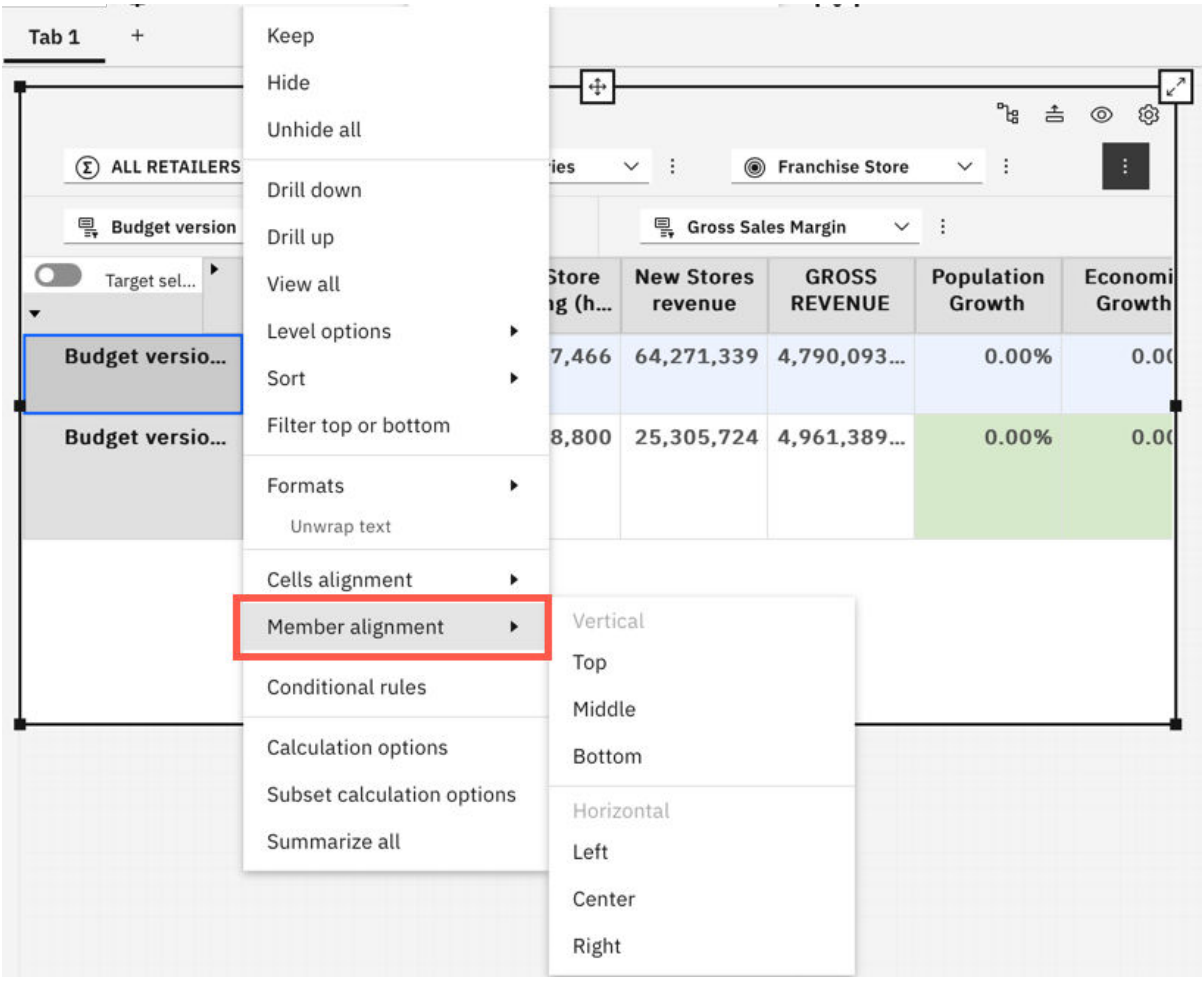
## Align cells and headers in an exploration

In Explorations, you can align text in data cells and headers either vertically or horizontally. You can align a row or column of data cells, align a member in row and column headers, and align based on hierarchy.

Right-click a data cell and select **Cells alignment** to see text alignment options.

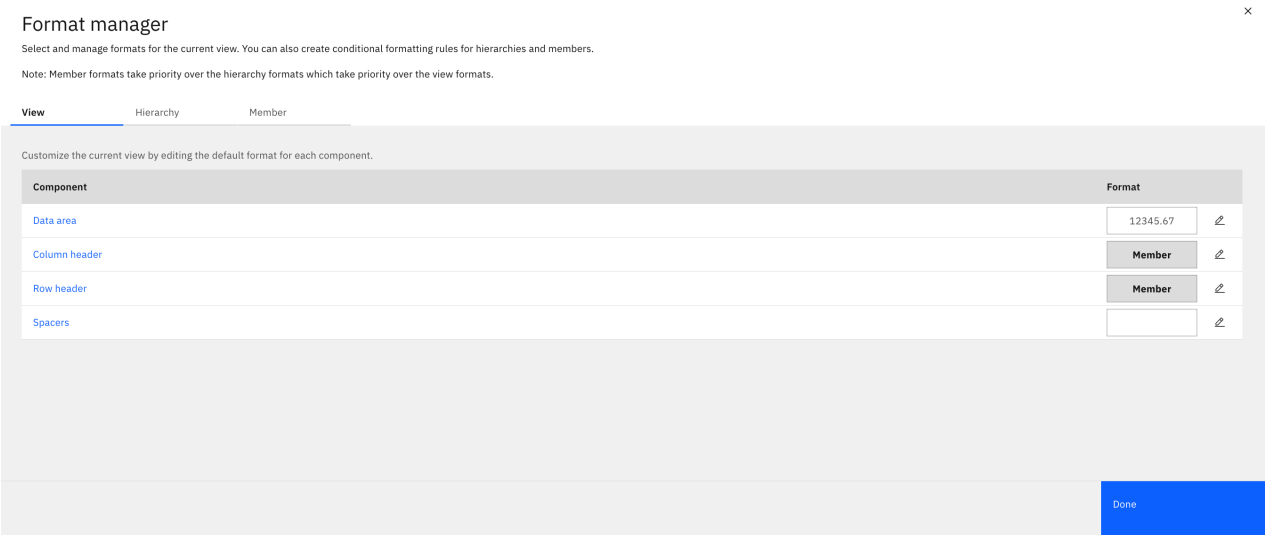
To align members in row or column headers, right-click a row or column header and under **Member alignment**, select an alignment option.


Tab 1
ALL RETAILERS
Budget version
Target sel...
Existing Stores Revenue
Budget versio...
4,725,821,663
Budget versio...
4,936,083,765
Franchise Store
Margin
GROSS REVENUE
Population Growth
Economi Growth
4,790,093...
0.00%
4,961,389...
0.00%
Keep
Hide
Unhide all
Drill down
Drill up
View all
Level options
Sort
Filter top or bottom
Formats
Unwrap text
Cells alignment
Member alignment
Conditional rules
Calculation options
Subset calculation options
Summarize all
Vertical
Top
Middle
Bottom
Horizontal
Left
Center
Right




## Format cube views with the Format manager

You can access formatting options for the cube viewer from the **Format manager**.




You can apply and manage the following formats for a view and the members in the current view:

**View**

- Cell color for data cells, row headers, and column headers
- Data formats for data cells

- Font and string formats for data cells, row headers, and column headers
- Background color and line dividers for spacer cells
- Thumb cells on the row and column axis of the cube viewer
- Customize the cell color by data state (rule-derived cells and read-only cells, and select font color for sandbox values)
- Customize the background and border colors of selected and highlighted cells
- Add **zebra strips** to apply a shaded background to alternating rows in an exploration

You can quickly apply common formats in the view with the **Inherit** toggle. When you set formats in a view, you can inherit formatting from data cells and apply it to row or column headers.

**Member**

- Data formats for data cells
- Font and string formats for data cells and header cells
- Member cell alignment
- Conditional rules for members

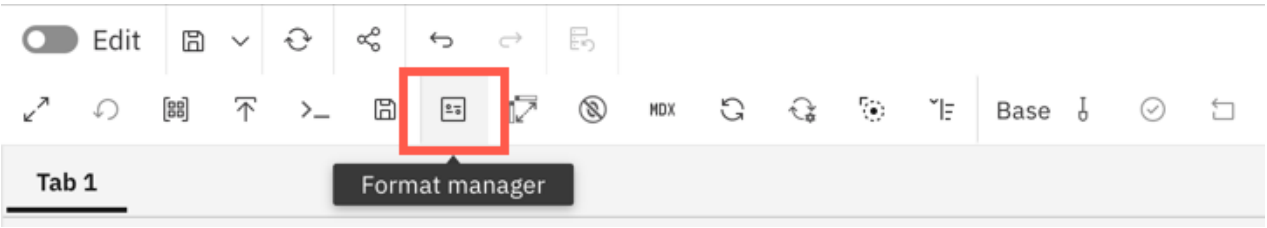
**Hierarchy**

- Cell color for data cells and header cells
- Font formats for data cells and header cells
- Cell alignment
- Conditional rules for hierarchy

**Note:** You can inherit formats and apply them to the member data or header cells.

## Accessing the Format manager

You can access the **Format manager** from the cube view toolbar.




Right-click a member in the view and select **Member** or **Hierarchy** to jump right into the member formatting options.

**Note:** You can no longer clear applied formatting from the right-click menu. Instead, use **Format manager** or **Member format** to clear or change formats.

You can manage the availability of **Format manager** through the **Toolbar** icon settings in the **Properties** panel. Disable **Format manager** under **Properties > Custom > Toolbar** if you don't want users to see the **Format manager** icon in the cube viewer toolbar.

## Setting hierarchy formats

To create a hierarchy formatting rule, follow these steps on the **Hierarchy** tab in the **Format manager**:

1. Select a dimension and hierarchy and click **Add format**.
2. Define the rule by selecting the condition type:
    a. **Base format** - Applies the rule to all of the members in the hierarchy.
    b. **Level** - Applies the rule to members of a selected hierarchy level.
    c. **Member attribute** - Applies the rule to members with a specific attribute value.

d. **Member name** - Applies the rule to members that contain a specific value in their name.

3. (Optional) Specify formats for the data cells and header cells.
4. Click **Apply** to save the format.

If you have member and view formats defined along with hierarchy formats, the formatting rules apply in the following order:

1. Member formats
2. Hierarchy formats on the Columns
3. Hierarchy formats on the Rows
4. View formats

If you have multiple hierarchy formatting rules, the first rule has the highest priority. You can rearrange the priority of a formatting rule by dragging it up or down the list of rules.

You can define a member format to have a specific background and font color and also use hierarchy formatting to define the cell and font colors based on the leaf level.

The following example uses hierarchy formatting to set the background color of the data and row headers based on their level. The leaf-level data values are displayed in gray font color.

| region | 1 Quarter | Jan | Feb | Mar | 2 Quarter | Apr | May | Jun |
|---|---|---|---|---|---|---|---|---|
| World | 21519.00 | 6713.00 | 7593.00 | 7213.00 | 16802.00 | 6409.00 | 5258.00 | 5135.00 |
| Europe | 17954.00 | 5759.00 | 6275.00 | 5920.00 | 13808.00 | 5286.00 | 4347.00 | 4175.00 |
| Scandinavia | 898.00 | 273.00 | 316.00 | 309.00 | 679.00 | 263.00 | 212.00 | 204.00 |
| Denmark | 248.00 | 76.00 | 87.00 | 85.00 | 189.00 | 74.00 | 59.00 | 56.00 |
| Norway | 293.00 | 87.00 | 104.00 | 102.00 | 230.00 | 88.00 | 72.00 | 70.00 |
| Sweden | 357.00 | 110.00 | 125.00 | 122.00 | 260.00 | 101.00 | 81.00 | 78.00 |
| Benelux | 719.00 | 336.00 | 382.00 | 1.00 | 819.00 | 308.00 | 262.00 | 249.00 |
| Belgium | 325.44 | 149.00 | 176.00 | 0.44 | 371.00 | 139.00 | 115.00 | 117.00 |
| Luxemburg | 33.05 | 16.00 | 17.00 | 0.05 | 37.00 | 14.00 | 12.00 | 11.00 |
| Netherlands | 360.51 | 171.00 | 189.00 | 0.51 | 411.00 | 155.00 | 135.00 | 121.00 |
| Islands | 2692.00 | 870.00 | 935.00 | 887.00 | 2034.00 | 773.00 | 646.00 | 615.00 |
| Great Britain | 2450.00 | 794.00 | 851.00 | 805.00 | 1849.00 | 701.00 | 588.00 | 560.00 |
| Ireland | 242.00 | 76.00 | 84.00 | 82.00 | 185.00 | 72.00 | 58.00 | 55.00 |
| Central Europe | 9446.00 | 2960.00 | 3166.00 | 3320.00 | 7143.00 | 2727.00 | 2231.00 | 2185.00 |
| France | 4503.00 | 1410.00 | 1526.00 | 1567.00 | 3334.00 | 1276.00 | 1025.00 | 1033.00 |
| Germany | 4943.00 | 1550.00 | 1640.00 | 1753.00 | 3809.00 | 1451.00 | 1206.00 | 1152.00 |
| Iberia | 1206.00 | 381.00 | 416.00 | 409.00 | 884.00 | 336.00 | 277.00 | 271.00 |
| Portugal | 201.00 | 63.00 | 70.00 | 68.00 | 147.00 | 55.00 | 48.00 | 44.00 |
| Spain | 1005.00 | 318.00 | 346.00 | 341.00 | 737.00 | 281.00 | 229.00 | 227.00 |

## Customizing cell color in views based on data state

Use the **Color cells by data state** property for views to customize a cell color based on its data state. You can pick a custom background color for rule-derived cells and read-only cells, and even customize the color of sandbox values. The **Color cells by data state** property can be found in the **Format manager**, under **Data area**.

**Important:** The **Color cells by data state** property overrides any other background color set for the cell.

To enable the **Color cells by data state** property for a view and customize the background color of cells, follow these steps:

1. Select a view and click **Format manager** in the toolbar.
2. Go to the **Data area** on the **View** tab.
3. Under **Table styles**, select **Color cells by data state** and choose the colors you want for **Rule-derived background**, **Read-only background**, and **Sandbox font**.
4. Click **Apply**.

**Note:** Rule-derived cells include cells with rule-derived values, calculations, and set consolidations. The **Color cells by data state**property does not apply to consolidated members.

## Customizing colors of selected and highlighted cells

Choose colors in the **Selection styles** section to customize the format of selected and highlighted cells. You can pick a custom color for the background and the border of the cells. You can also select a transparent background color, which allows you to view the original color of the active cells without overwriting the colors.

Customize the color of the background and the border of any cell that you select and any highlighted cell that is associated with the selected cells. If you select the background color to be transparent, selecting or highlighting the cells does not change the original background color of these cells.

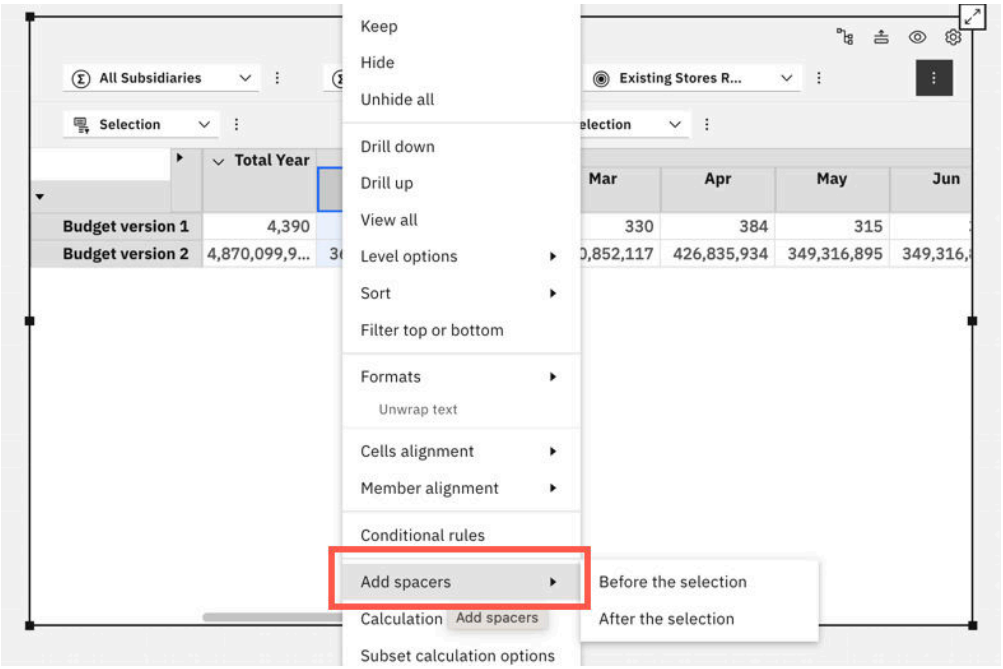
To customize the format of selected and highlighted cells, follow these steps:

1. Select a view and click **Format manager** in the toolbar.
2. Click **Data area** on the **View** tab.
3. Under **Selection styles**, choose the colors you want for **Selected background**, **Selected border**, **Highlighted background**, and **Highlighted border**.

   To keep the original cell colors of the selected and highlighted cells, select transparent (no color) as the color for **Selected background** and **Highlighted background**.
4. Click **Apply**.

## Insert spacer rows and columns in explorations

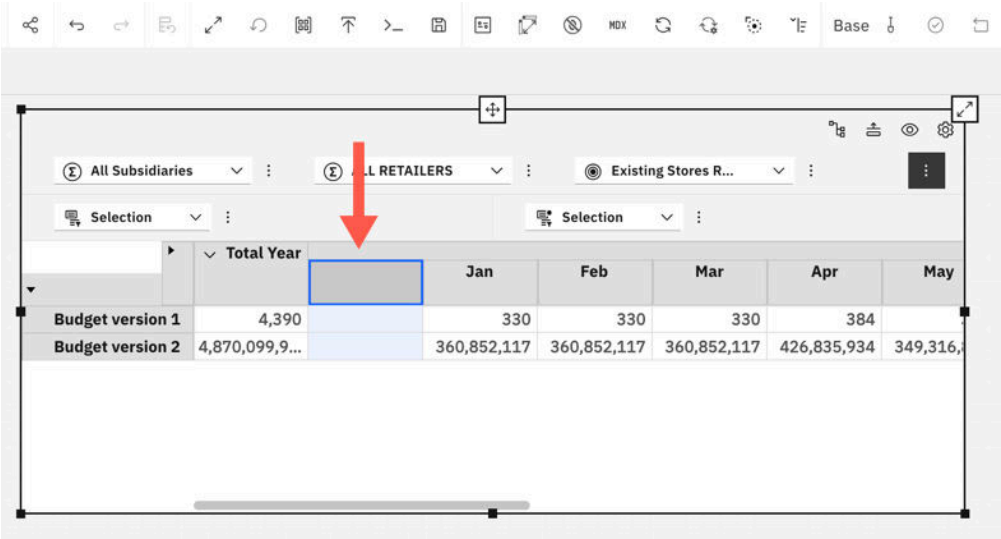
You can insert spacer rows and columns in explorations through the right-click menu. This is useful for building stylized views of data in Books.




To insert a spacer:

1. Select a row or column and right-click a row or column header in an exploration.
2. Click **Add spacers** and select where you want to insert the spacer row or column.

A spacer row or column is inserted at the position you selected.




You can copy and paste a row or column with spacer cells. However, the spacer cells can break some spreads.

To remove a spacer row or column, right-click the spacer header cell and select **Remove**.

Click **Format manager** to show and hide the line dividers for spacer cells and to set the background color for spacer cells. You can also modify existing spacer cells, by right-clicking the spacer cells in a view and selecting **Spacer format**.

# Analyze views with AI

You can use the following features to analyze views with AI.

## Outlier analysis

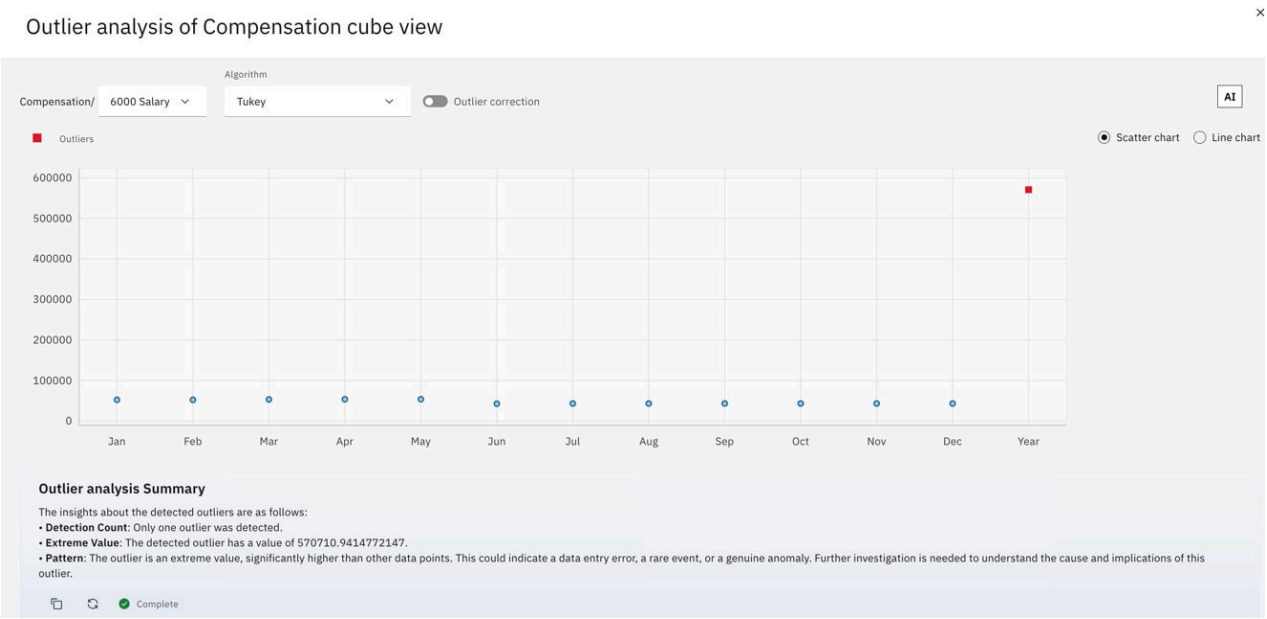
Use **Outlier analysis** to identify and correct data points in an exploration that deviate significantly from the norm, potentially indicating errors or anomalies. The analysis also includes AI-generated insights for any outliers that were detected and corrected in the view, helping you to investigate anomalies and improve the quality of your data.

The initial release of the **Outlier analysis** feature is available only in Planning Analytics Workspace Cloud and in English only.

**Important:** The **Outlier analysis** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

To perform an outlier analysis, click an exploration in a book and select the rows that you want to analyze. Then, click **AI operations** in the toolbar and select **Outlier analysis**.

You can select up to 25 rows to run **Outlier analysis** but the feature considers only the first 120 columns in the exploration for the analysis.




By default, the resulting analysis displays the data in the form of a Scatter chart, making it easy to see any outliers. The algorithm selected to conduct the outlier analysis is evaluated by using the Silhouette score and the best ranked algorithm is then used to run the analysis. However, you can select a different algorithm and regenerate the analysis. You can also switch the chart type from a Scatter chart to a Line chart.

To correct the outliers, enable **Outlier correction**. The visualization adjusts by using a “normalized” data point for the outliers.

**Important:** Outlier correction does not modify the actual data point, that was identified as an outlier, on your Planning Analytics database.

An AI-generated summary of the outlier analysis automatically generates when you run the feature. If you have **Outlier correction** enabled, the summary includes insights for the outliers that were corrected.

## Algorithms used in outlier analysis

**Tukey's method**
: Tukey's method identifies outliers by calculating the interquartile range (IQR) and defining lower and upper bounds. Data points outside these bounds are considered outliers.

**Z-Score method**
: The Z-Score method is a statistical technique that standardizes data points to determine how many standard deviations they are from the mean. A Z-score greater than a specified threshold (commonly 3) indicates an outlier.

**One-Class Support Vector Machine (SVM)**
: One-Class SVM is a supervised machine learning algorithm that learns a decision boundary around the majority class (normal instances of the data points). Data points outside this boundary are classified as outliers.

**Isolation Forest**
: Isolation Forest builds an ensemble of decision trees to isolate data points. Outliers are detected as points that require fewer splits to isolate than the majority of data points.

**Local Outlier Factor (LOF)**
: LOF algorithm measures the local density deviation of a given data point with respect to its neighbors. Data points with a lower density than their neighbors are classified as outliers.

**DBSCAN**
: DBSCAN is a density-based clustering algorithm that groups data points based on their density. Data points in low-density regions that do not belong to any cluster are considered outliers.

**Ensemble Voting method**

The Ensemble Voting method combines the results of the various outlier detection algorithms to improve the robustness of the detection process. By using a voting mechanism, this method selects the most frequently identified outliers across all methods, enhancing overall accuracy. This approach is effective when different algorithms yield varying results, as it uses the strengths of each method.

The effectiveness of each outlier detection method is evaluated with **Silhouette score**. This score measures how similar an object is to its own cluster compared to other clusters, providing a way to assess the quality of clustering and separation of outliers. A higher Silhouette score indicates better-defined clusters.

For datasets with fewer than 50 data points, only Tukey's and Z-Score methods are applied. More complex algorithms, such as One-Class SVM, Isolation Forest, LOF, and DBSCAN, require a larger dataset to provide reliable results and accurately capture data distributions.

### AI-generated summary

The **Outlier analysis** feature uses granite-3-8b-instruct from the Granite family of IBM foundation models to generate the summary.

The Granite models are decoder-only models that can efficiently predict and generate language. These models were built with trusted data that has the following characteristics:

- Sourced from quality data sets in domains such as finance (SEC Filings), law (Free Law), technology (Stack Exchange), science (arXiv, DeepMind Mathematics), literature (Project Gutenberg (PG-19)), and more.
- Compliant with rigorous IBM data clearance and governance standards.
- Scrubbed of hate, abuse, and profanity, data duplication, and blocklisted URLs, among other things.

**Note:** IBM is committed to building AI that is open, trusted, targeted, and empowering. For more information about contractual protections that are related to IBM indemnification, see the IBM Client Relationship Agreement and IBM watsonx.ai service description.

### Tips on working with AI

While working with **Outlier analysis**, here are few things that you can do that might lead the AI model to generate a better analysis:

- Ensure that the values in the exploration data are numeric.
- Remove spacers from rows or columns.
- Remove duplicate values.

## Impact analysis

Use **Impact analysis** on an exploration to determine the impact of data fields on each other as well as identify and quantify relationships across the data. Generative AI interprets and summarizes the analysis to describe the direct and indirect impact of the data fields on each other, as well as any dependencies and relationships between them.

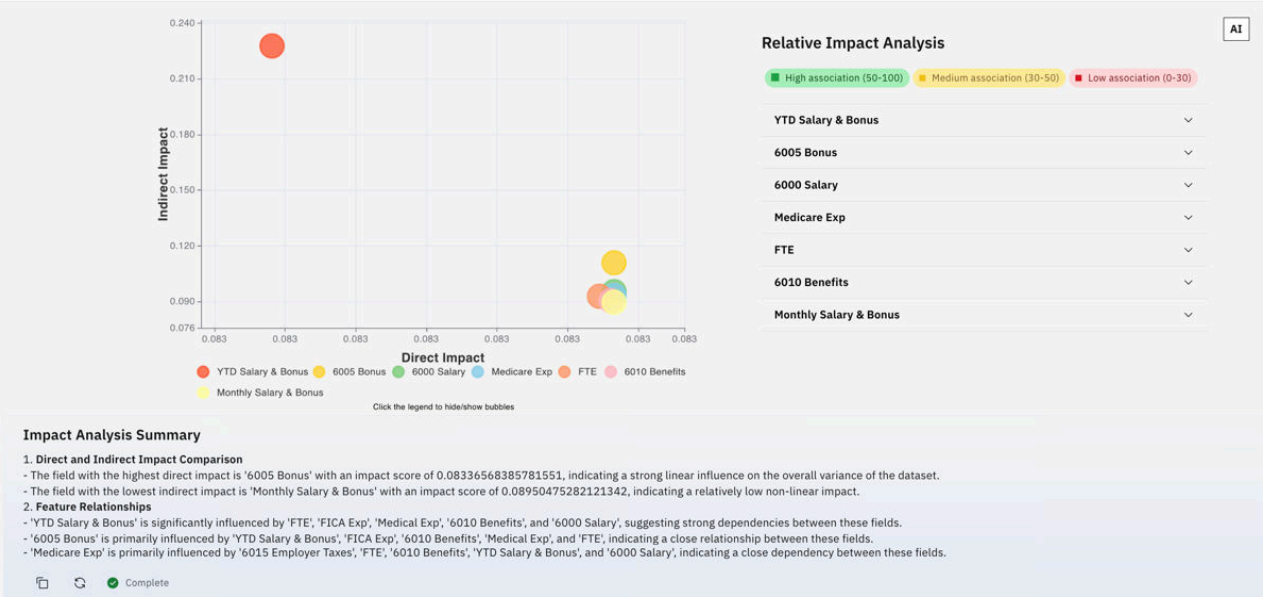
The initial release of **Impact analysis** is available only in Planning Analytics Workspace Cloud and in English only.

**Important:** The **Impact analysis** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

To start an **Impact analysis**, click an exploration in a book. Then, click **AI operations** in the toolbar and select **Impact analysis**. To run the analysis, the exploration must have a minimum of two rows and six columns. The analysis is run on the first 120 rows and columns of the view.

The completed analysis results in a Bubble chart, which visually depicts the relationships between the data fields, along with an AI-generated summary of the analysis.

Impact Analysis of Compensation cube view




You can hover over a bubble to see the data field name and its direct and indirect impact values. You can also click the labels in the legend to show or hide the related bubbles in the chart.

When you click a bubble in the chart, the related section under **Relative impact analysis** expands to display the relative contribution of data fields on the selected data. The **Relative impact analysis** quantifies the degree of association between two data fields so that you can view the data that drives or influences the selected data.

## Techniques used in impact analysis

Two techniques, Linear Principal Component Analysis (PCA) and Kernel PCA, are used to identify and extract key features from the data. Scores are compared from both methods to gain insight into the underlying structure of the data and the importance of individual features.

**Principal Component Analysis (PCA)**

PCA is a dimensionality reduction technique that transforms high-dimensional data into a lower-dimensional space while retaining as much variance as possible. This is crucial in data analysis as it helps in:

- Reducing complexity by simplifying datasets to make them easier to visualize and interpret.
- Removing noise by filtering out less informative features, which can improve model performance.
- Uncovering patterns by revealing relationships and patterns that might not be evident in high-dimensional data.
- Unsupervised learning as PCA and Kernel PCA are both unsupervised techniques and they analyze the data without any reference to target labels. This allows for a more objective exploration of the data's structure and relationships among features.

Linear PCA operates under the assumption that the data is linearly separable and identifies the directions (principal components) in which the data varies the most. The scores from Linear PCA represent the projection of the original data onto these principal components. Each score indicates how much variance a feature contributes to the overall data structure. A higher score signifies that a feature is more influential in determining the position of the data point in the reduced space.

Kernel PCA extends the capabilities of Linear PCA by applying a kernel function to project data into a higher-dimensional space, capturing nonlinear relationships. This makes Kernel PCA useful

for datasets where the relationships between features are not linear. Kernel PCA scores reveal the importance of features, but they also account for complex interactions and nonlinearity in the data. A high Kernel PCA score suggests that a feature plays a significant role in the structure of the data in the transformed space.

**Feature scores**

Understanding the scores from both Linear and Kernel PCA is important for:

- Feature selection: Identifying which features to keep or discard based on their importance.
- Model optimization: Improving machine learning models by focusing on the most informative features.
- Data interpretation: Gaining insights into the data structure and feature interactions, which can guide further analysis and decision-making.

**Visualization of scores**

To compare the results of Linear PCA and Kernel PCA, scores from Linear PCA are plotted on the x-axis and the scores from Kernel PCA, on the y-axis. This visual representation helps in assessing the relationship between the two methods.

Points in different areas of the Bubble chart can indicate how features perform under linear versus nonlinear transformations.

- Upper right area: Features with high scores in both Linear and Kernel PCA indicate strong relevance and contribution to the data structure.
- Bottom left area: Features with low scores in both methods might be less informative.
- Cross-area scores: Features that score high in one method but low in another might warrant further investigation to understand their unique contributions.

Relative impact analysis is achieved by using Gradient Booster Regressor. This technique identifies contributor features related to a specific target variable in a dataset. Gradient Boosting is an ensemble learning technique that constructs a model in a stage-wise fashion, optimizing for accuracy by combining the predictions of multiple weak learners. This approach not only enhances prediction capabilities but also provides essential insights into feature importance, helping to identify the features that significantly impact the target variable.

## AI-generated summary

The **Impact analysis** feature uses granite-3-8b-instruct from the Granite family of IBM foundation models to generate the summary.

The Granite models are decoder-only models that can efficiently predict and generate language. These models were built with trusted data that has the following characteristics:

- Sourced from quality data sets in domains such as finance (SEC Filings), law (Free Law), technology (Stack Exchange), science (arXiv, DeepMind Mathematics), literature (Project Gutenberg (PG-19)), and more.
- Compliant with rigorous IBM data clearance and governance standards.
- Scrubbed of hate, abuse, and profanity, data duplication, and blocklisted URLs, among other things.

**Note:** IBM is committed to building AI that is open, trusted, targeted, and empowering. For more information about contractual protections that are related to IBM indemnification, see the IBM Client Relationship Agreement and IBM watsonx.ai service description.

## Tips on working with AI

While working with **Impact analysis**, here are few things that you can do that might lead the AI model to generate a better analysis:

- Ensure that the values in the exploration data are numeric.
- Remove spacers from rows or columns.

- Remove duplicate values.

# Exploration analysis

The **Exploration analysis report** feature uses AI to generate an analysis of a selected view in the form of a PDF report. The generated report includes an executive summary followed by an impact analysis and outlier analysis for the exploration, allowing you to take faster business decisions.

The initial release of **Exploration analysis report** feature is available only in Planning Analytics Workspace Cloud and in English only.

**Important:** The **Exploration analysis report** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM® sales representative or go directly to your IBM account.

To start an Exploration analysis, follow these steps:

1. Select an exploration in a book.
2. In the toolbar, click **AI operations** and then select **Exploration analysis report**.
3. After the report is generated, a notification displays from which you can open the PDF file in the finder or open it in a new tab.

The Exploration analysis is run on the first 120 rows and columns of the exploration.

When you run an **Exploration analysis report**, an impact analysis and outlier analysis is performed on the exploration.

The impact analysis determines the impact of data fields on each other as well as identifies and quantifies relationships across the data. Generative AI interprets and summarizes the analysis to describe the direct and indirect impact of the data fields on each other, as well as any dependencies and relationships between them.

An outlier analysis on the exploration identifies data points that deviate significantly from the norm, potentially indicating errors or anomalies. AI-generated insights are available for any outliers that were detected, helping you to investigate anomalies and improve the quality of your data.

## AI-generated summary

The **Exploration analysis report** feature uses granite-3-8b-instruct from the Granite family of IBM foundation models to generate the executive summary and insights from the impact and outlier analysis.

The Granite models are decoder-only models that can efficiently predict and generate language. These models were built with trusted data that has the following characteristics:

- Sourced from quality data sets in domains such as finance (SEC Filings), law (Free Law), technology (Stack Exchange), science (arXiv, DeepMind Mathematics), literature (Project Gutenberg (PG-19)), and more.
- Compliant with rigorous IBM data clearance and governance standards.
- Scrubbed of hate, abuse, and profanity, data duplication, and blocklisted URLs, among other things.

**Note:** IBM is committed to building AI that is open, trusted, targeted, and empowering. For more information about contractual protections that are related to IBM indemnification, see the IBM Client Relationship Agreement and IBM watsonx.ai service description.

## Tips on working with AI

While working with **Exploration analysis report**, here are few things that you can do that might lead the AI model to generate a better analysis:

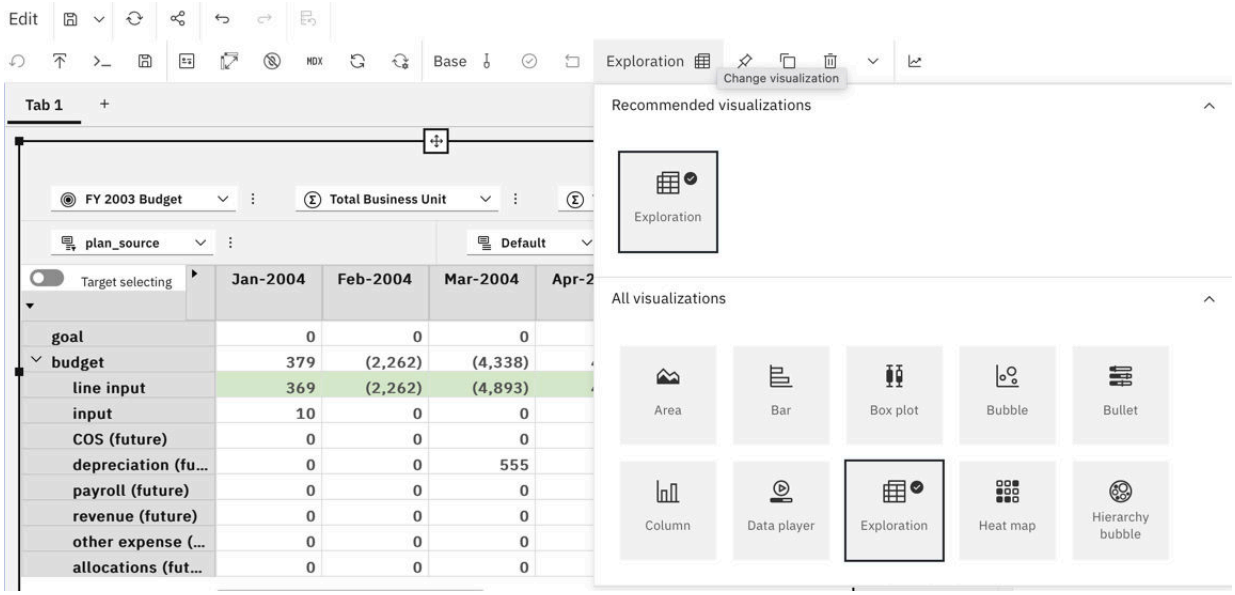
- Ensure that the values in the exploration data are numeric.
- Remove spacers from rows or columns.
- Remove duplicate values.

# Visualizations in Planning Analytics Workspace

You can use any of the visualizations that are described here to present data in Planning Analytics Workspace.

To change the visualization type for any view in a book:

1. Click the view.
2. Click the **Change visualization** icon. The label next to the icon indicates the type of visualization that is currently displayed for the view.




3. Under **All visualizations**, select the visualization that you want to apply to the view.

**Note:** When you convert a view to a visualization, if Planning Analytics Workspace cannot find a mapped member, the first member in the current set is automatically reassigned to the missing mapped member's field slot in order to display the selected visualization. This can occur if a mapped member was removed from the data as a result of the following:

- A change to the set (dynamic or static) such as a synchronization event
- View actions such as zero suppression, calculation removal, and more
- Insufficient permissions in the view. For example, if User1 creates a view that includes members for which User2 does not have at least Read permission.

You might still see a blank visualization if :

- a substitute member is not available
- synchronization forces the visualization to use the missing mapped member
- the field slot is reassigned to a substitute member but you force the visualization to use the missing mapped member by selecting it from the member selector menu

You can also drag a cube or a view from the data tree onto an existing visualization to replace the visualization. The visualization updates based on the view you drop.

**Note:** Replacing the visualization resets any previous visualization properties and fields that you manually selected or added.

When you right-click a visualization member, you see the full abbreviated value of the member, along with several options to interact with the visualization.

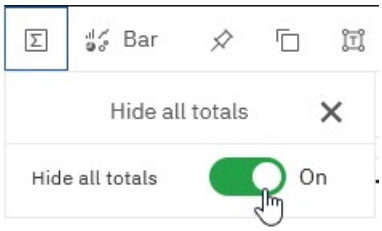
**Hide**
Hide the member in the visualization.

**Unhide all**
Reveal all hidden members in the visualization.

**Drill up**
Collapse the member to show its immediate parent.

**Drill down**
Reveal the members of a consolidation. You cannot drill down on a leaf member.

To hide totals in a visualization, click the **Hide all totals** button on the toolbar, then turn on the **Hide all totals** option.




To restore totals in the visualization, turn off the **Hide all totals** option.

# Create a visualization directly in a book

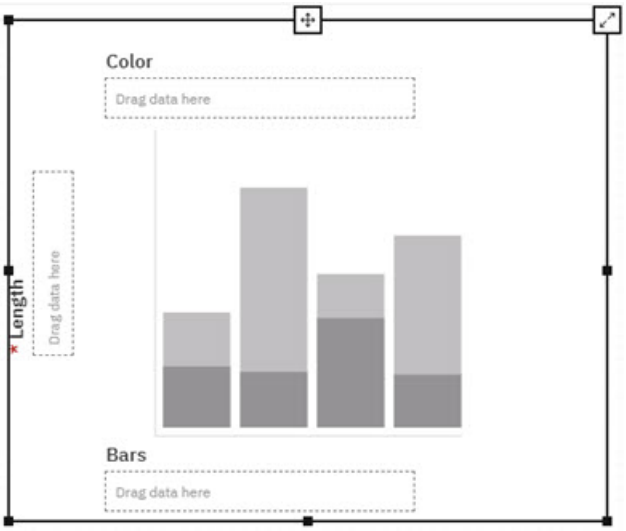
You can build a visualization from scratch in a book, without having to convert an existing Exploration.

## About this task

You must be logged in as an analyst, modeler, or administrator to create a visualization.

## Procedure

1. Create a new book or open the book where you want to create the visualization.
2. Enable Edit mode.
3. Click the **Visualizations** button, then click the type of visualization you want to create.
   A template for your visualization is inserted into your book.



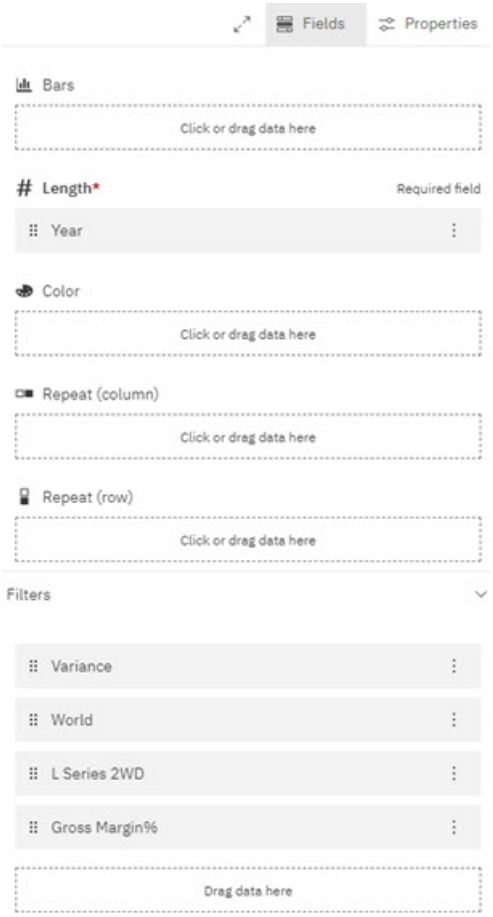

The template shows an example of the type of visualization you are building. There are several **Drag data here** drop zones representing fields in your visualization. Required fields are identified by an asterisk.

For illustration purposes, we'll build a **Stacked column** visualization, but the principles described here apply to all visualizations.

4. Click the **Database** icon to open the Data tree.
5. Locate the cube that you want to use as the data source for the visualization.
6. Expand the cube and identify the dimension that you want to insert in one of the required fields of the visualization. Alternatively, you can identify a subset or single member to insert into the visualization.
7. Drag and drop the dimension (or subset or member) onto the **Drag data here** drop zone for a required field.

   At this point, you've established a relationship between the cube and the visualization. You can now use the other dimensions of the cube to complete the visualization.

   Take a look at the **Fields** tab and you'll notice that the dimension that you dropped onto the visualization is in the appropriate field. All other dimensions from the cube are positioned as Filters. Filters set the context for a visualization, just as dimensions on the bench determine the context for an Exploration.



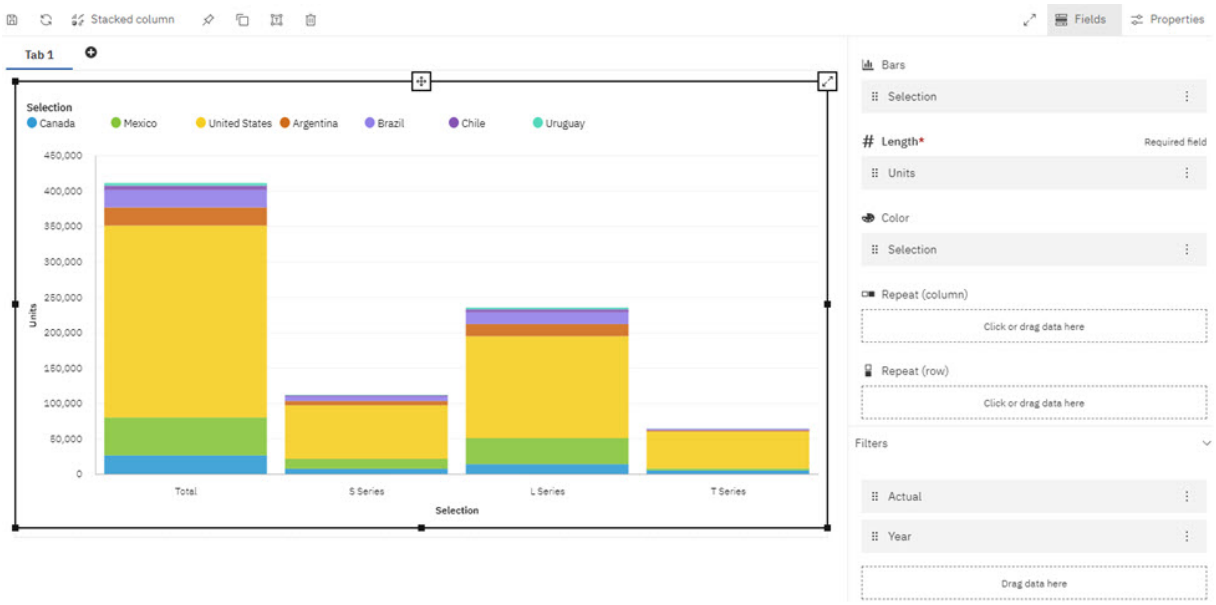

   You can finish building your visualization directly in the Fields tab.
8. Drag and drop a dimension from the **Filters** list onto one of the fields for the visualization.

   You'll immediately see the impact of your action in the visualization. Continue moving dimensions from the Filters list to the visualization fields until your visualization is configured the way you want it. At a minimum, you have to define the required fields for the visualization, but the more fields you define the greater the detail you can see in the visualization.

   You can ignore the **Repeat (row)** and **Repeat (column)** fields for now, we'll discuss those later.

   You can modify the visualization configuration by dragging and dropping dimensions from one field to another. When you drop a dimension on an occupied field, the dimension positions are switched. For example, if you drag *DimensionA* from the Color field onto *DimensionB* on the Bars field, you end up with *DimensionA* on the Bars field and *DimensionB* on the Color field.

Here's an example of a fully configured Stacked column visualization. You can see that all fields other than **Repeat (row)** and **Repeat (column)** are defined.

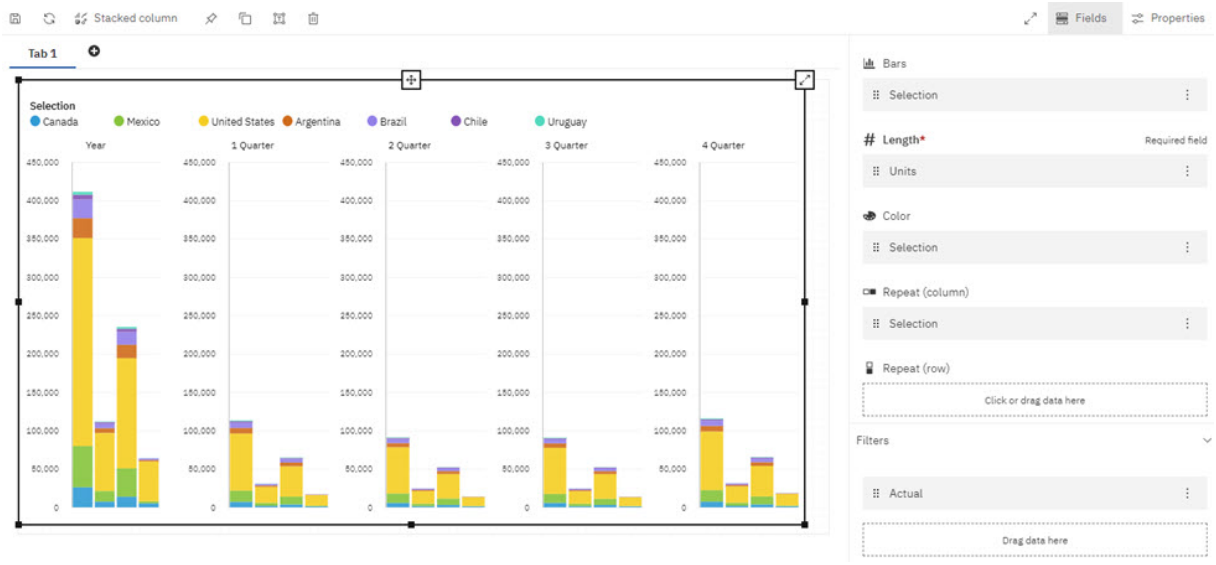



9. Click on any occupied field to further refine your visualization.

   When you click on an occupied field, including fields in the Filters list, you can select a different member from the current set or a defined dimension level. You can also click **Edit this set** to open the Set Editor and modify the set as desired.

10. Now that you've got your visualization dialed in, you can use the **Repeat (row)** or **Repeat (column)** fields to repeat the visualization for multiple members. When you drag a dimension to the **Repeat (row)** or **Repeat (column)** field, the visualization is repeated for each member in the dimension set, oriented along either rows or columns.

    For example, the Stacked column visualization shows information for the entire year, but you might want to repeat the visualization for each quarter in the year. In this case, you'd drag Year from the Filters list to the **Repeat (column)** field. You might have to modify the set slightly, but the result is a repetition of the visualization for each quarter.




## Visualizations available in Planning Analytics Workspace

The following visualizations can be created in Planning Analytics Workspace.

## Area

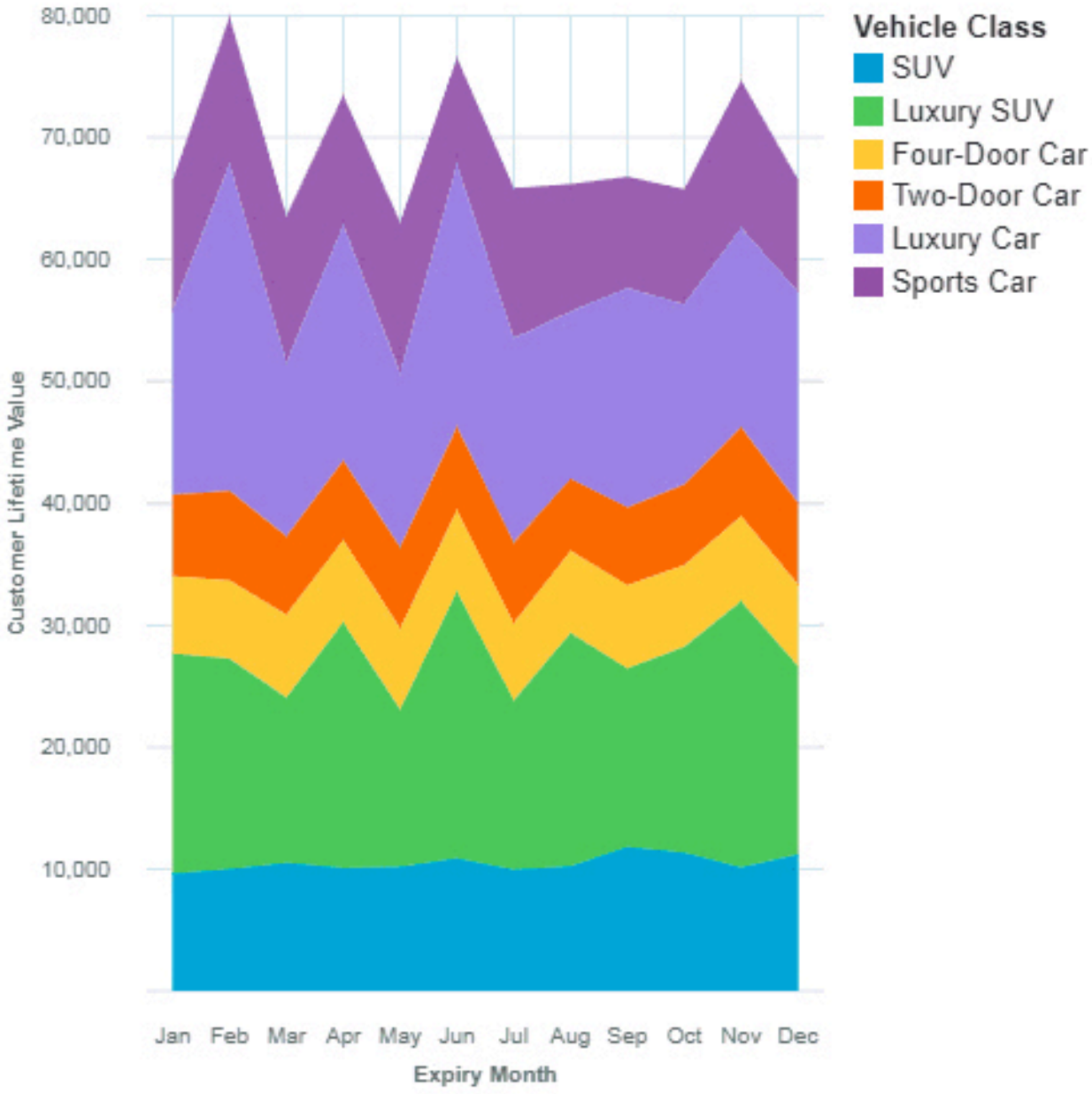
Use an area visualization to emphasize the magnitude of change over time.

Area charts are like line charts, but the areas below the lines are filled with colors or patterns. Stacked charts are useful for comparing proportional contributions in a category. They plot the relative value that each data series contributes to the total.

Because an area visualization stacks the results for each column or item, the total of all results is easily seen.

For example, an area visualization is excellent for looking at revenue over time across several products.

This area visualization shows the customer lifetime value for each vehicle class per month. Because the area visualization stacks the results, you see the totals for each month.

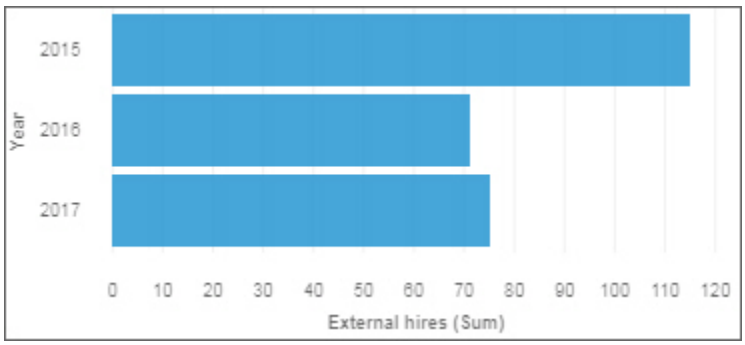


## Bar

Use a bar visualization to compare values by one or more columns, such as sales for products or sales for products each month.

Bar visualizations use horizontal data markers that are arranged in groups to compare individual values. You can use bar visualizations to compare discrete data or to show trends over time.

A bar visualization can show change over a specific time period or can compare and contrast two or more columns in a time period or over time. If there are so many bars that the labels are impossible to read, filter the data to focus on a subset of the data or use a tree map.




Use the **Target** field to show measures that need to be compared against a target value.

Use the **y-start** field to define where the measure must start.

## Box plot

You can use box plot visualizations to identify outliers and compare distributions.

You can create a box plot to show the median, quartiles, and outlier and extreme values for a variable. The inter-quartile range is the difference between the 75th and 25th percentiles and corresponds to the length of the box. The middle line is the 50th percentile.

Above and under each box, whiskers give additional information about the spread of the data.

Far out values are represent by adding "o" signs beyond the whiskers.

The mean score in a box plot is presented by a "+" sign.

Use the Key field in a box plot visualization to determine the items for which you want to identify outliers and compare distributions.

This box plot example shows the gross profit statistics for various markets.



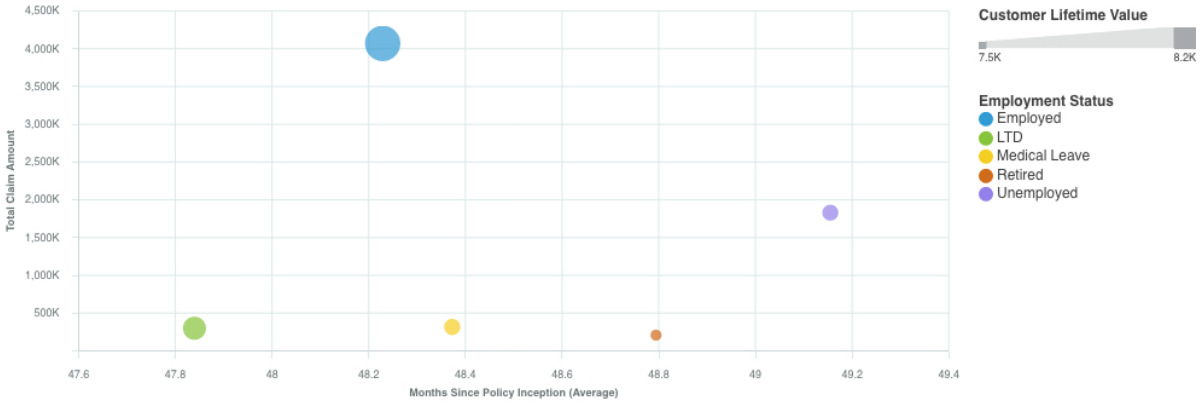

## Bubble

Use a bubble visualization to show relationships among columns that contain numeric values, such as revenue and profit.

A bubble visualization uses data points and bubbles to plot measures anywhere along a scale. One measure is plotted along each axis. The size of the bubble represents a third measure. Use bubble visualizations to represent financial data or any data where measure values are related.

The bubbles are in different sizes and colors. The x-axis represents one measure. The y-axis represents another measure, and the size of the bubbles represents the third measure. In the following example, color is represented by an identifier.

The example that is shown represents the months since the policy inception.




## Bullet

Use bullet charts to show measures that need to be compared against a target value.

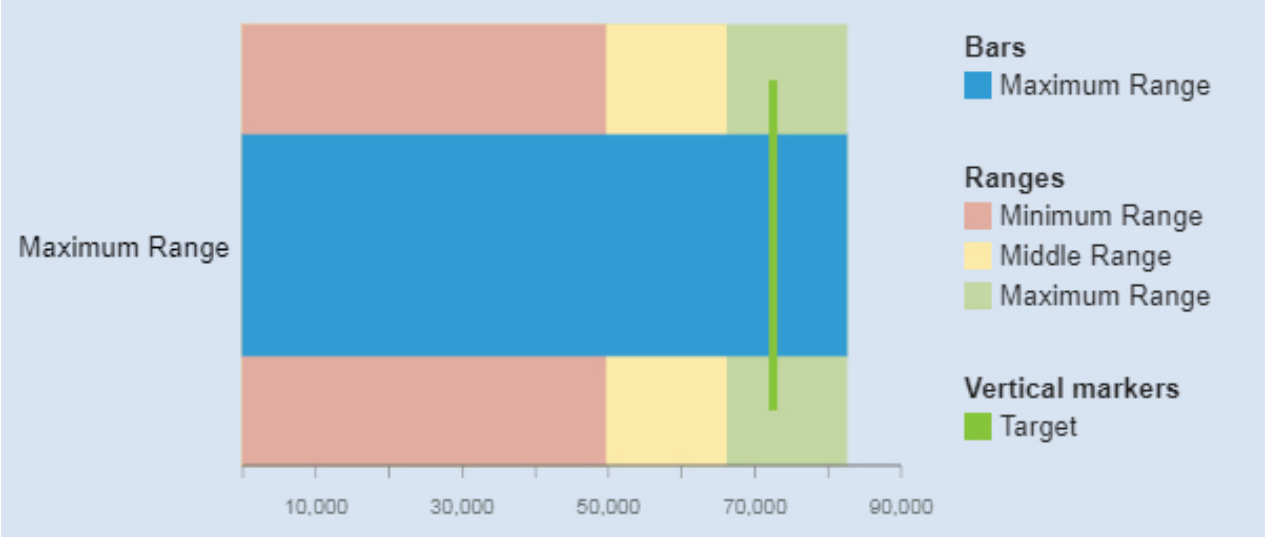
In a call center, a Bullet visualization can be used to measure metrics like call volume, call answer speed, and percentage of abandoned calls.

In manufacturing, a Bullet visualization can be used to track metrics like number of defects and orders that are shipped.

In a fitness context, a Bullet visualization can be used to measure metrics like steps that are taken and calories that are burnt.

Bullet visualizations compare an actual measure (the bullet) to targeted measure (the target). Bullet visualizations relate the compared measures against colored regions in the background that provide more qualitative measurements, such as good, satisfactory, and poor. Bullet visualizations can be shown at small sizes while still effectively conveying information.

A Bullet visualization features a single, primary measure (such as current year-to-date revenue) and compares that measure to one or more other measures (such as a target) for enhanced analysis. The primary measure displays in the context of a qualitative range of performance, such as poor, satisfactory, and good.




When you create a Bullet visualization, specify the following fields under **Fields**:

- The **Actual bar** field specifies the actual measure.
- The **Target** field specifies the target measure.
- The **Minimum range** field specifies the minimum qualitative range.
- The **Middle range** field specifies the middle qualitative range.
- The **Maximum range** field specifies the higher qualitative range.

Make sure the minimum, medium, and maximum ranges relate to the actual and target measure.

Use the various properties under **Properties** to style the visualization. For example, you can style gridlines and axes lines, style axes titles and labels, add padding to the bullet, target, or ranges, and much more.

**Creating multiple Bullet visualizations in a widget**

You can stack multiple Bullet visualizations in one widget by using the **Repeat** slots or the **Categories** slot under **Fields**.

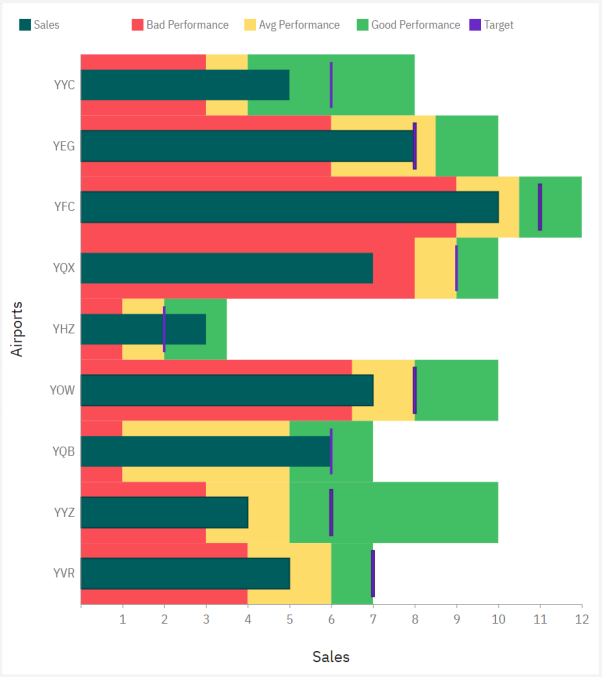
Prior to Planning Analytics Workspace 2.0.87, you could only use the **Repeat** slots to create multiple Bullet visualizations. As of Planning Analytics Workspace 2.0.87, you can use **Categories** slot to draw multiple bullets without the use of **Repeat** slots.

Bullet visualizations created with the **Categories** slot feature an improved design and an easier-to-read display. In the following example, notice that in Figure B all of the Bullet visualizations share the axes.

**Figure A: Repeat slots used to create multiple Bullet charts**




**Figure B: Categories slot used to create multiple Bullet charts**




When creating multiple Bullet charts, it is recommended to use the **Categories** slot.

Apart from better readability, using the **Categories** slot allows additional sorting capabilities such as sorting by Target and Bar values and enhanced legend interactivity. Hover over the qualitative range legend labels to see the number of members that are in that range. To view all the members in a specific range, click the qualitative range legend labels.

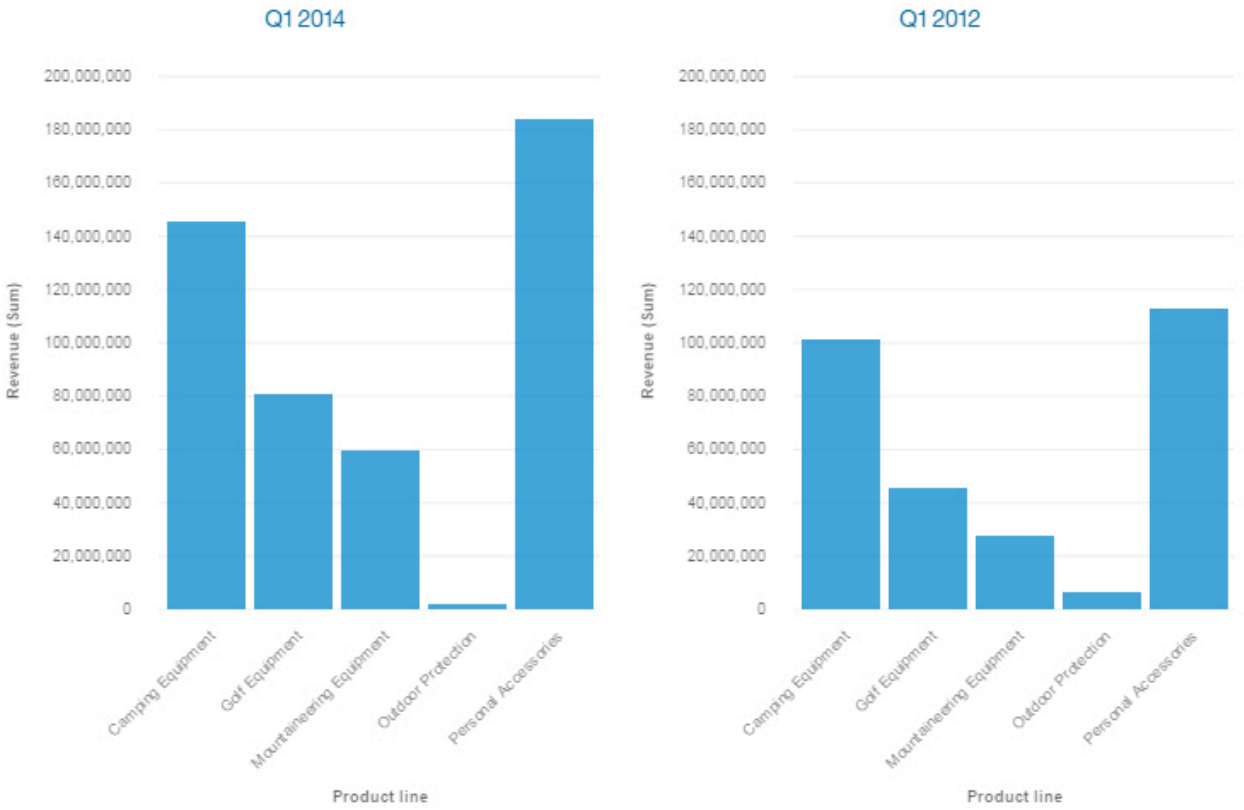
## Column

Use a column visualization to compare values by one or more columns, such as sales for products or sales for products each month.

Column visualizations use vertical data markers that are arranged in groups to compare individual values. Use column visualizations to compare discrete data or show trends over time.

A column visualization shows change over a specific time period or can compare and contrast two or more columns in a time period or over time. If there are so many bars that the labels are impossible to read, filter the data to focus on a subset of the data or use a tree map.

For example, revenue for each product line is grouped by quarter, which emphasizes performance in each quarter.




Use the **Target** field to show measures that need to be compared against a target value.

Use the **y-start** field to define where the measure must start.

## Data player

Use a data player to see an animation of the impact of a column on the other visualizations.

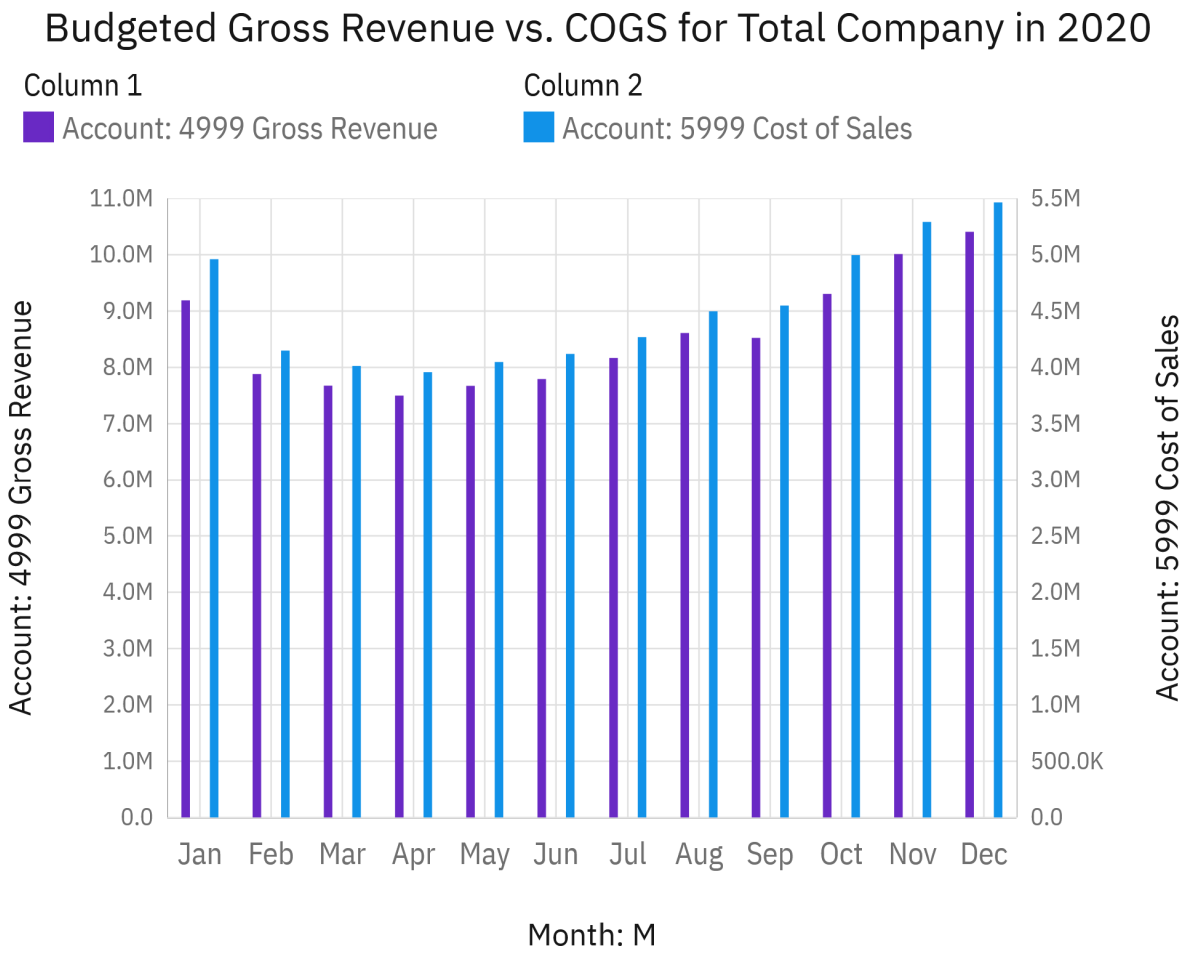
## Dual axes column

Use a dual-axis column visualization to display relationships between two data sets. Each series of columns in the visualization represents one y-axis.

This visualization technique allows to display two different data sets in one visualization. Each y-axis uses its own scale. Using this technique, you can compare different metrics or measures that each have different scales or units. When users view the data side-by-side, it is easier for them to understand patters and correlations in the data.

A dual-axis line visualization is similar to the “Line and column” on page 495 visualization. Both visualizations show how two data sets relate to one another. The difference is that the dual-axis column visualization presents both data sets as columns, while the line and column visualization uses columns for one data set and a line for the second data set.

The following image shows an example of a dual-axis column visualization.



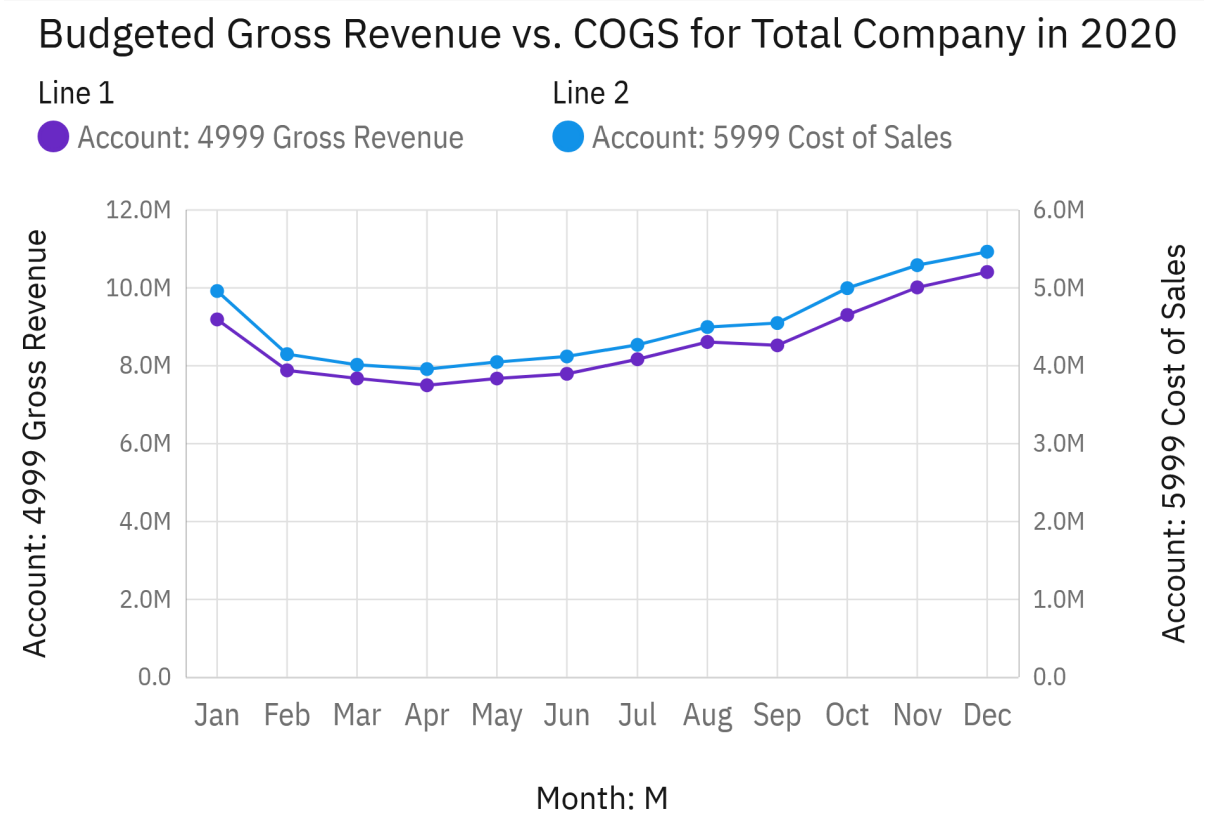

## Dual axes line

Use a dual-axis line visualization to display relationships between two data sets. Each line in the visualization represents one y-axis.

This visualization technique allows you to display two different data sets in one visualization. Each y-axis uses its own scale. Using this technique, you can compare different metrics or measures that each have different scales or units. When users view the data side-by-side, it is easier for them to understand patters and correlations in the data.

A dual-axis line visualization is similar to the “Line and column” on page 495 visualization. Both visualizations show how two data sets relate to one another. The difference is that the dual-axis line visualization presents both data sets as lines, while the line and column visualization uses a line for one data set and columns for the second data set.

The following image shows an example of a dual-axis line visualization.

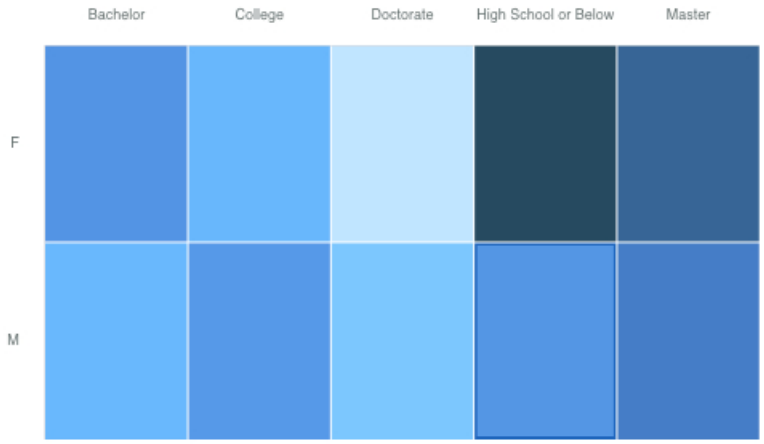



## Heat map

Use a heat map visualization to visualize the relationship between columns, represented in a matrix type view.

A heat map visualization uses color and intensity of the color to show the relationship between two columns.

For example, this heat map visualization shows the average customer lifetime value by gender and education.




## Hierarchy bubble

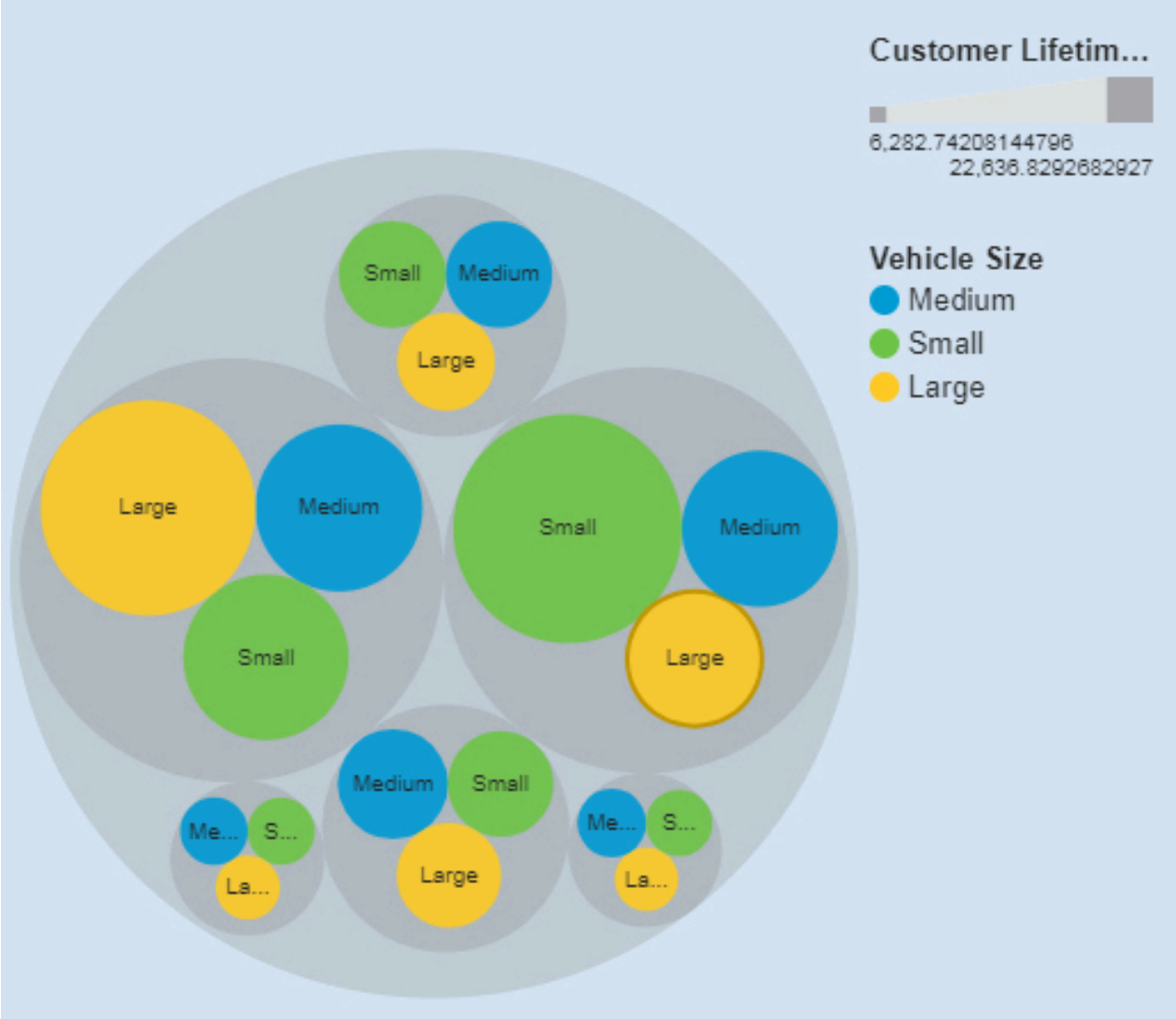
Use a hierarchy bubble visualization when you want to show relationships among columns that contain values, such as net loss. It is similar to the bubble visualization but the bubbles are tightly packed instead

of spread over a grid. The bubbles use nesting to represent the hierarchy. A hierarchy bubble visualization shows a large amount of data in a small space.

The size of each bubble shows a quantitative dimension of each data point. It shows many levels within a hierarchy and relationships between groups based on assigned attributes. It uses bubble size and color to convey comparative information about categories.

The bubbles are in different sizes and colors.

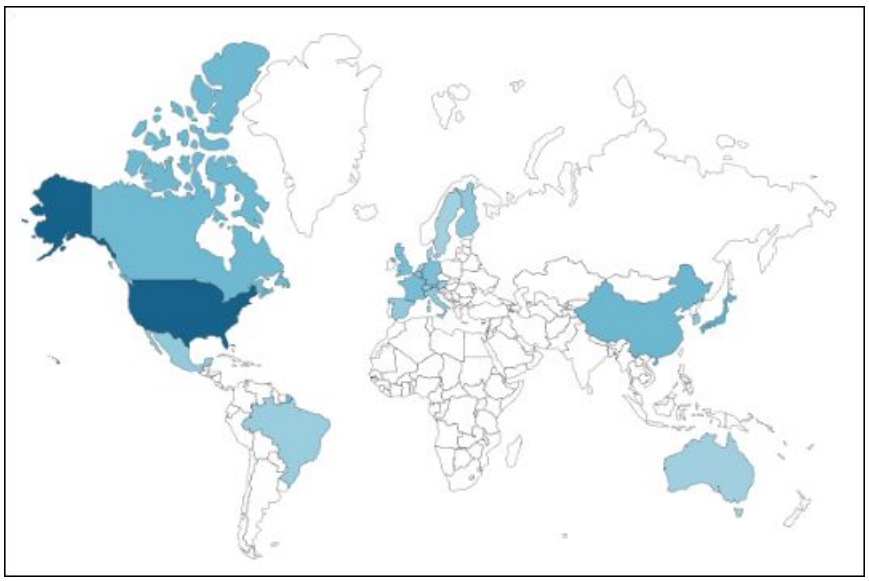
For example, this hierarchy bubble visualization shows customer lifetime value by vehicle class per vehicle size. Each bubble is a different vehicle class in one of the three vehicle size. The size of each bubble is determined by the customer lifetime value of that vehicle class. The colors of the bubbles are determined by the vehicle size.




## Legacy map

Use a legacy map when you want to see patterns in your data by geography.

For example, this legacy map visualization shows revenue by retailer country with the darker color indicating higher revenue.



To create a map visualization, your view must contain a defined geography dimension.

The geography dimension must contain elements that are recognizable as geographic entities. To determine if a dimension is mappable, Planning Analytics Workspace analyzes a sample of members in the defined geography dimension, looking for recognizable place names. If 80% or more of the members in the geography dimension are recognized as geographic entities, a map is generated.

For more information on the languages and geographic entities supported in map visualizations, see "Map reference info" on page 1267.

## Legacy waterfall

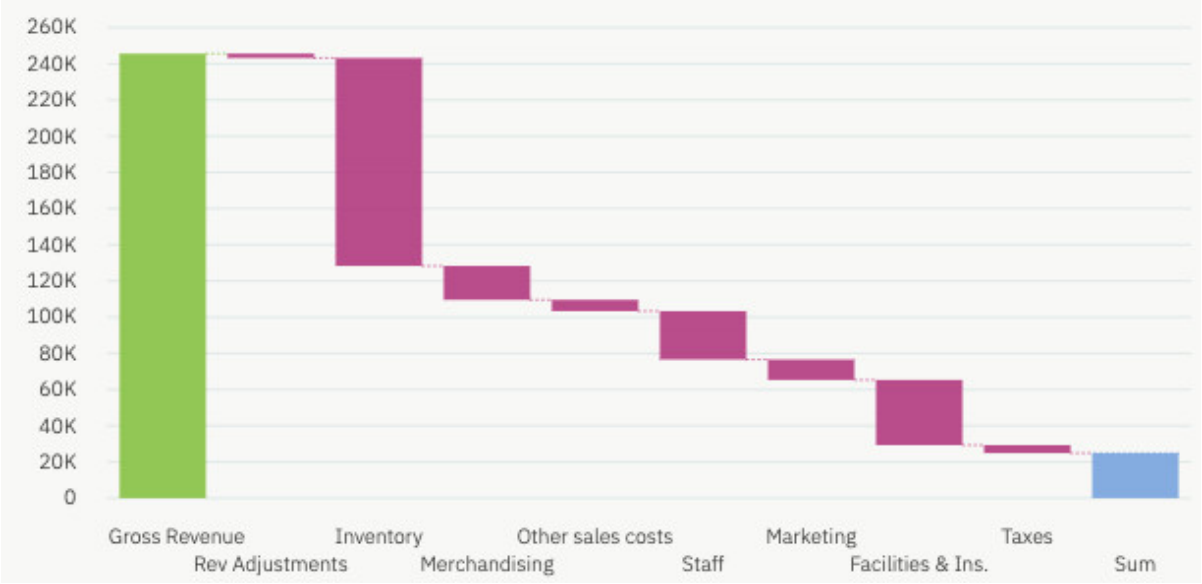
Use a waterfall visualization to understand the cumulative effect a series of positive and negative values have on an initial value. The bars in a waterfall visualization are not totals.

A waterfall visualization shows how an initial value is increased and decreased by a series of intermediate values, leading to a final cumulative value shown in the far right column. The intermediate values can either be time-based or category-based.

Some examples of waterfall visualizations are as follows:

- Viewing the net income after you add the increases and decreases of revenue and costs for an enterprise over a quarter.
- Cumulative sales for products across a year with an annual total.

This waterfall visualization shows the policy holder delta by month.



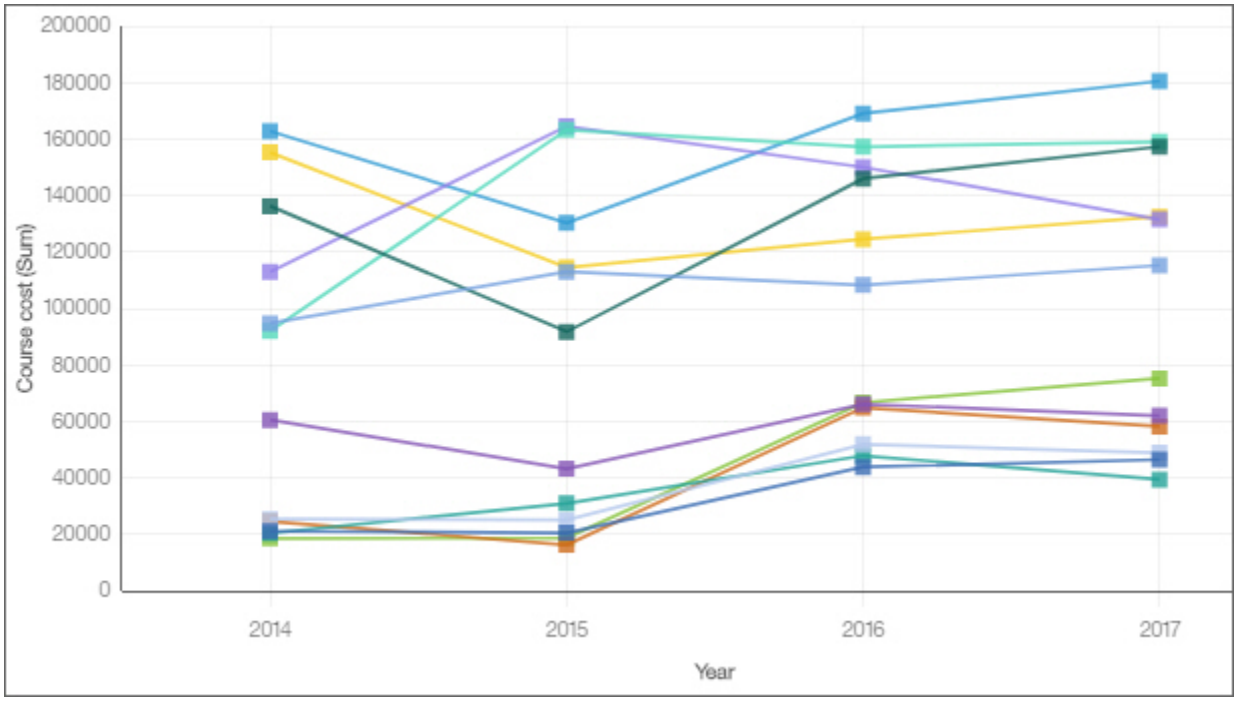

## Line

Use a line visualization to show trends over time.

A line visualization can compare trends and cycles, infer relationships between variables, or show how a single variable is performing over time.

For an effective line visualization, use a time column in the x-axis, such as years, quarters, months, or days. If the x-axis shows something else, such as Canada, Netherlands, UK, and US, use a bar or column visualization.

For example, this line visualization shows the trend in course costs by department over year.



### View forecast values in a line visualization

If a forecast has previously been generated for a view, you can configure a line visualization to display forecast values. for the view. To learn more about how to generate a forecast, see “Forecasting” on page 526.

To display forecast data in a line visualization:

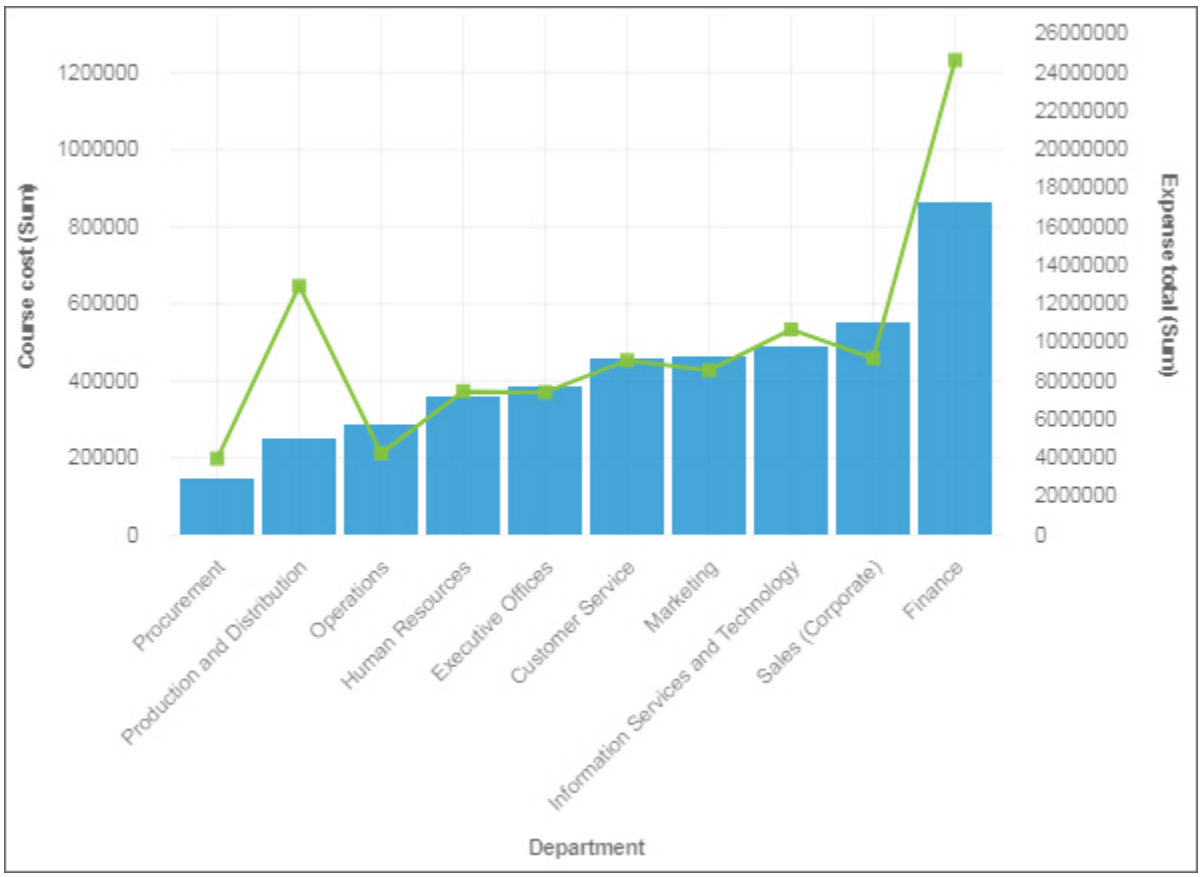
1. Configure your visualization with the time dimension on the x-axis and the version dimension as a context dimension.
2. Click the **Fields** tab.
3. Expand the **Forecast fields** section.
4. Click the **Forecast start** field and select a starting period for the forecast.
5. Click the **Dimension** field and select the version dimension for your view.
6. Click the **Hierarchy** field and select the version hierarchy for the forecast.
7. Click the **Forecast values** field and select the version dimension member that stores forecast values. This member name was set when you initially generated the forecast, and will typically be something like `Forecast`.
8. Click the **High bound values** field and select the version dimension member that stores high bound forecast values. These values are the high bound of the confidence envelope for a forecast. This member name was set when you initially generated the forecast, and will typically be something like `Forecast high`.
9. Click the **Low bound values** field and select the version dimension member that stores low bound forecast values. These values are the low bound of the confidence envelope for a forecast. This member name was set when you initially generated the forecast, and will typically be something like `Forecast low`.

The line visualization displays forecast values along a broken line that extends from the non-forecast values. You can hover on any forecast data point to view the forecast value, along with the upper and lower bound values. Click any data point to reveal the confidence envelope for the forecast.

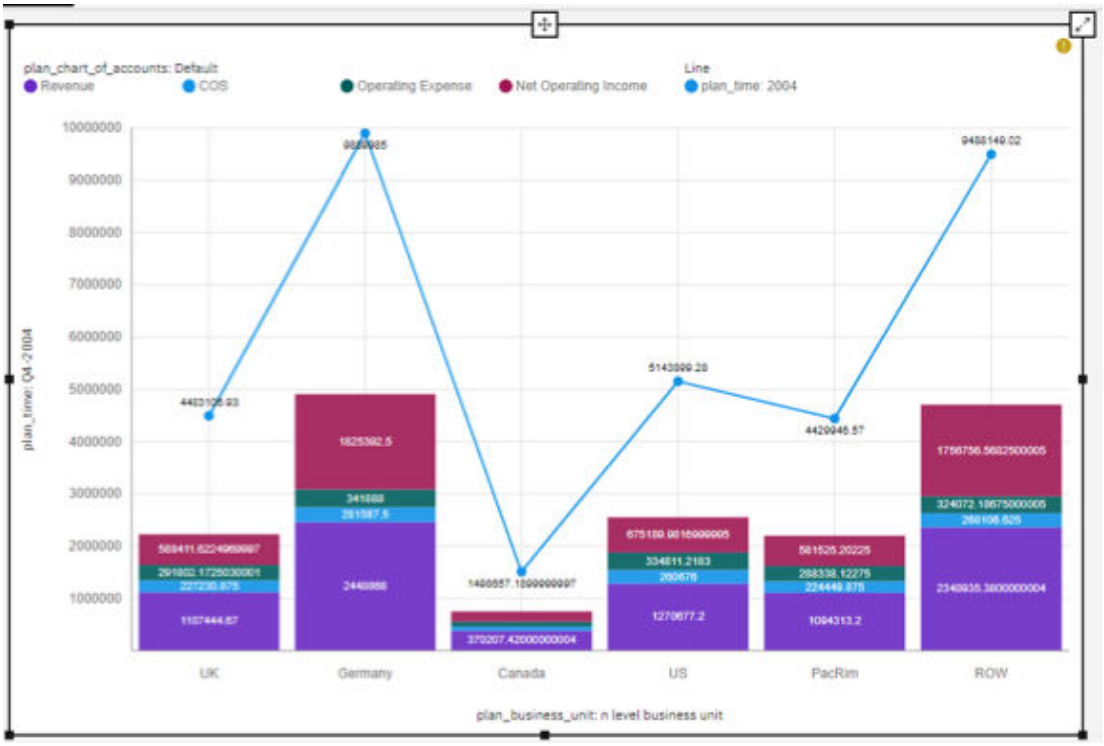
## Line and column

Use a line and column visualization to highlight relationships between multiple data series by combining bars and lines with one visualization.

For example, this line and column visualization shows the relationship between course cost and expense totals by department.

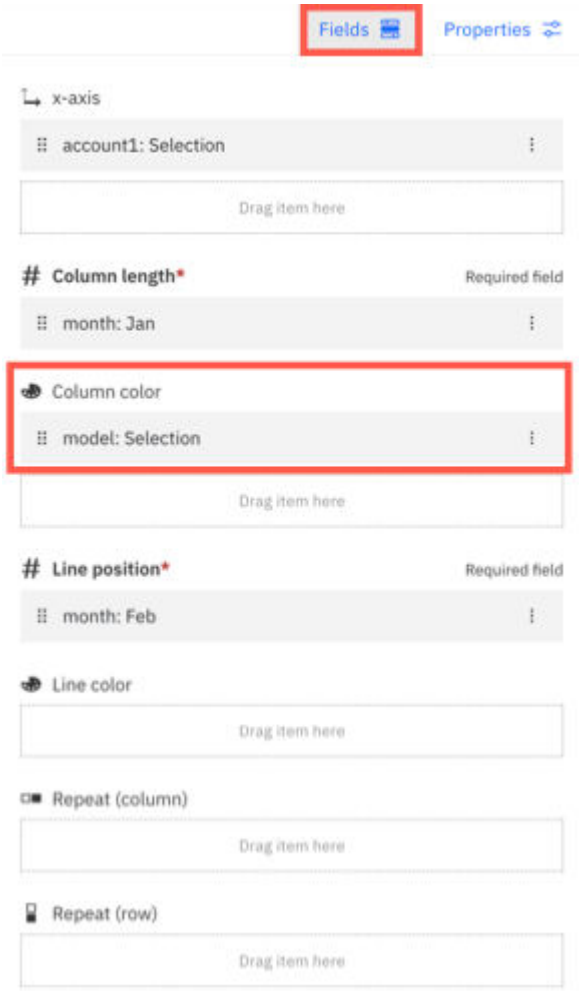



You can also create a combination chart with stacked columns using the line and column chart.



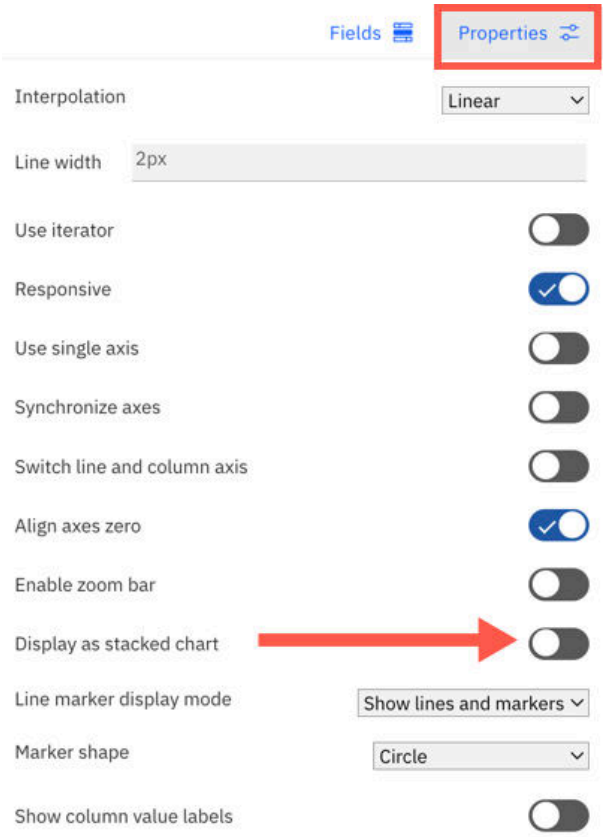

To create a combination stacked column chart, first create a line and column visualization and then follow these steps:

1. In **Edit** mode, select the line and column visualization that you just created and go to **Fields**.
2. Set the **Column color** field.




3. Go to **Properties** and under **Visualizations**, expand the **Chart** section.
4. Turn on the **Display as stacked chart** property to display the chart as a combination stacked column chart.

   **Note:** The **Display as stacked chart** property displays only when the **Column color** field is set.




## List

Use a list visualization to create an overview of data in an hierarchical way.

## Map

Use a Map when you want to see patterns in your data by geography. Data is mapped by matching the location name on the map with a location in your data.

A map must reference at least one dimension with geographic members, such as countries, states, provinces, or continents.

The geography dimension must contain members that are recognizable as geographic entities. To determine whether a dimension is mappable, Planning Analytics Workspace analyzes a sample of members in the defined geography dimension, looking for recognizable place names. If 80% or more of the members in the geography dimension are recognized as geographic entities, a map is generated.

For more information on the languages and geographic entities supported in map visualizations, see Map reference info.

You can also integrate Mapbox with Map visualization to display custom polygons, icons, and layers. For more information, see “Setting up Mapbox to work with Planning Analytics Workspace Map visualizations” on page 500.

### Map properties

You can customize a Map visualization using properties under **Properties > Visualization properties**. Here are some key properties you can use:

#### World View

The **World View** property displays disputed country borders from the perspective of different countries.

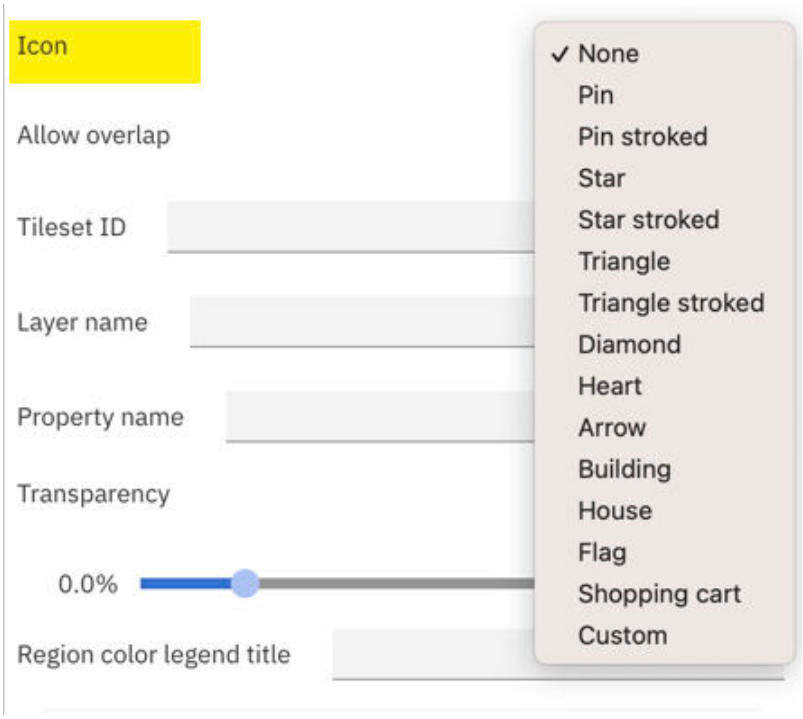
In the **World View** property under **Chart**, select a country and notice how the visualization changes the way disputed country borders display based on the selected country's perspective.

By default, the **World View** property is set to United States (US). Other countries you can choose from are:

- IN - India
- CN - China
- JP - Japan
- AR - Argentina
- MA - Morocco
- RU - Russia
- TR - Turkey

**Icons**

You can select a pre-defined icon for points on a Map. Depending on whether you want to add icons to the regions or points layer of the Map, expand either the **Regions layer** section or **Points layer** section. Select an icon from the **Icon** property dropdown.




You can also select a custom icon.

**Note:** When you choose to use a custom icon, you no longer have access to the predefined icons.

To use a custom icon in Maps, make sure you set up Mapbox to work with IBM Planning Analytics Workspace. Then, follow these steps:

1. In Mapbox Studio, create a new style using a template.
2. Customize the selected template.
3. Under **Layers**, pick the layer for your Map visualization.
4. Under **Icon**, pick your custom icon. You can also upload an image that you want to use as an icon.

   **Tip:** Make a note of the name of the custom icon you picked or added. You will need to add the custom icon name to the visualization properties in Planning Analytics Workspace later.
5. Publish the style.

6. Go to the style you just created and copy the style URL.
7. In Planning Analytics Workspace, click the visualization and go to **Properties > Visualization properties > Chart**.
8. In the **Map base** dropdown, select **Custom**.
9. Paste the style URL in the **URL** field.
10. Expand the **Points layer** section and in the **Icon** dropdown, select **Custom**.
11. Enter the name of the custom icon in **Custom icon name** field.

**Allow overlap**

Use the **Allow overlap** if you want icons to display on each point even if another element (such as point labels) is conflicting with the icons.

You can hide the default circle icon for points in the Map by setting the **Transparency** property to **100%**.

**Show map labels**

Turn on the **Show map labels** under **Chart** to see background map labels.

### *Setting up Mapbox to work with Planning Analytics Workspace Map visualizations*

The Map visualization in IBM Planning Analytics Workspace supports the use of custom polygons, icons, and layers that come from Mapbox.

To get started, create a free Mapbox account. When you have created your Mapbox account, upload your GeoJSON custom polygon file to Mapbox as a tileset. If your GeoJSON is less than 5 MB, then you can edit the GeoJSON directly as a data set in Mapbox. After you edit the GeoJSON files, export them into a tileset. Make sure that each polygon has at least one uniquely identifying property as a string value. For more information about tilesets, see the Mapbox documentation about a tileset.

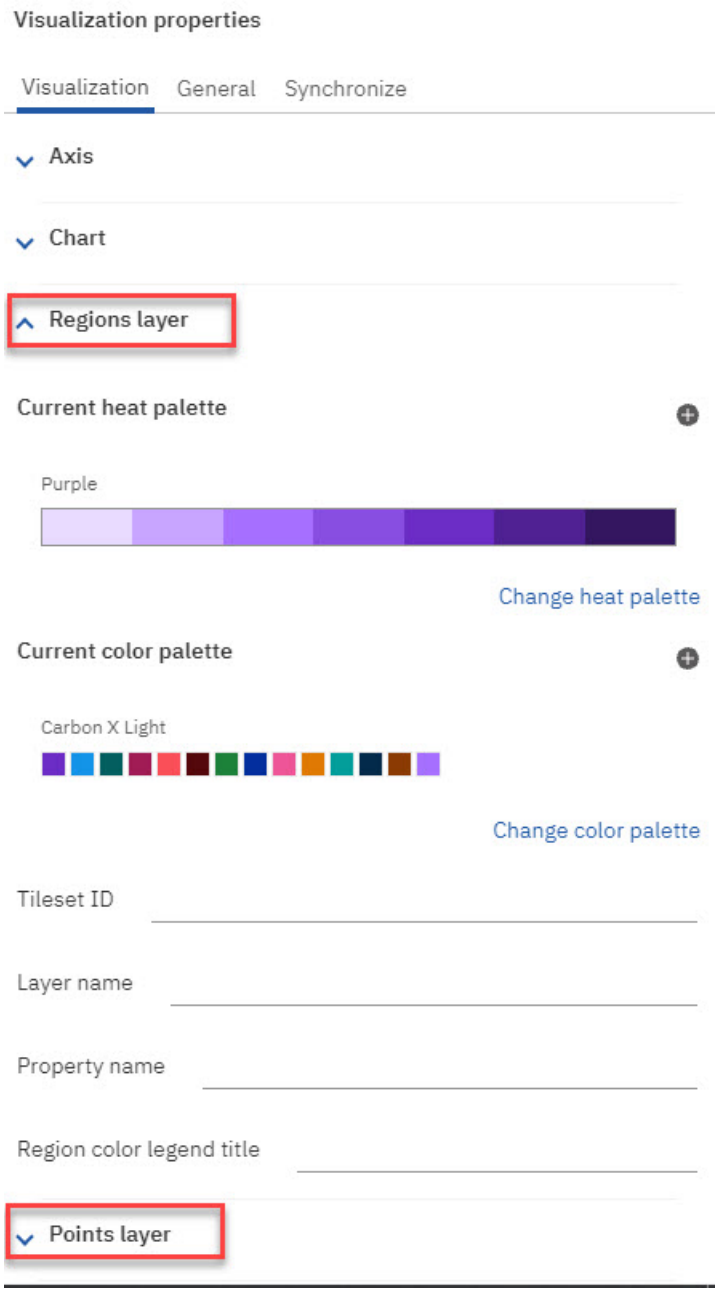
Custom polygons in formats such as .shp and KML can be converted to GeoJSON, by using, for example QGIS.

## Retrieving data from Mapbox

When you have your tileset created in Mapbox, make note of the following information from the tileset page:

- Tileset Id (appears in the **Share & use** panel)
- Layer name (appears in the **Vector layers** panel)
- Property name (appears in the **name** field for the layer)

You need to specify these values as **Regions layer** or **Points layer** properties of the Map visualization when you want to use Mapbox data in your map.




## Best practices

- For increased performance and to make sure that polygons appear at zoom level 0 (world view), compress geoJSON files to less than 10 MB. A tool that can compress geoJSON files is http://mapshaper.org/.

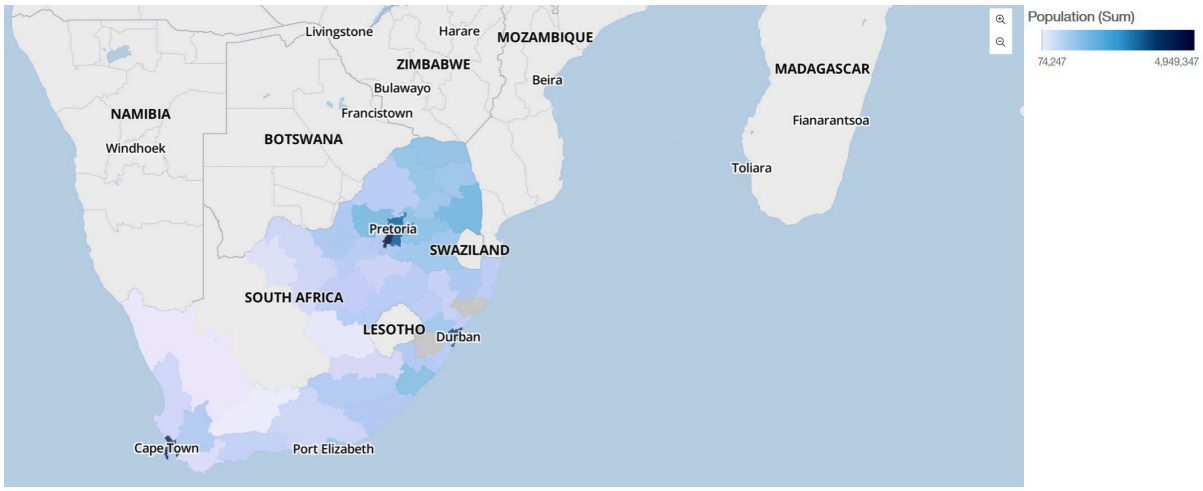
  If your geoJSON files are larger than 10 MB, then Mapbox sets the default zoom to not start at worldview 0. If you want to manually change the zoom extents, there are few ways to do so:

  – Tippecanoe APIs, which are available on Linux and Apple MacOs.
  – Download Mapbox studio classic on your Microsoft Windows computer and change the minimum and maximum zoom levels.

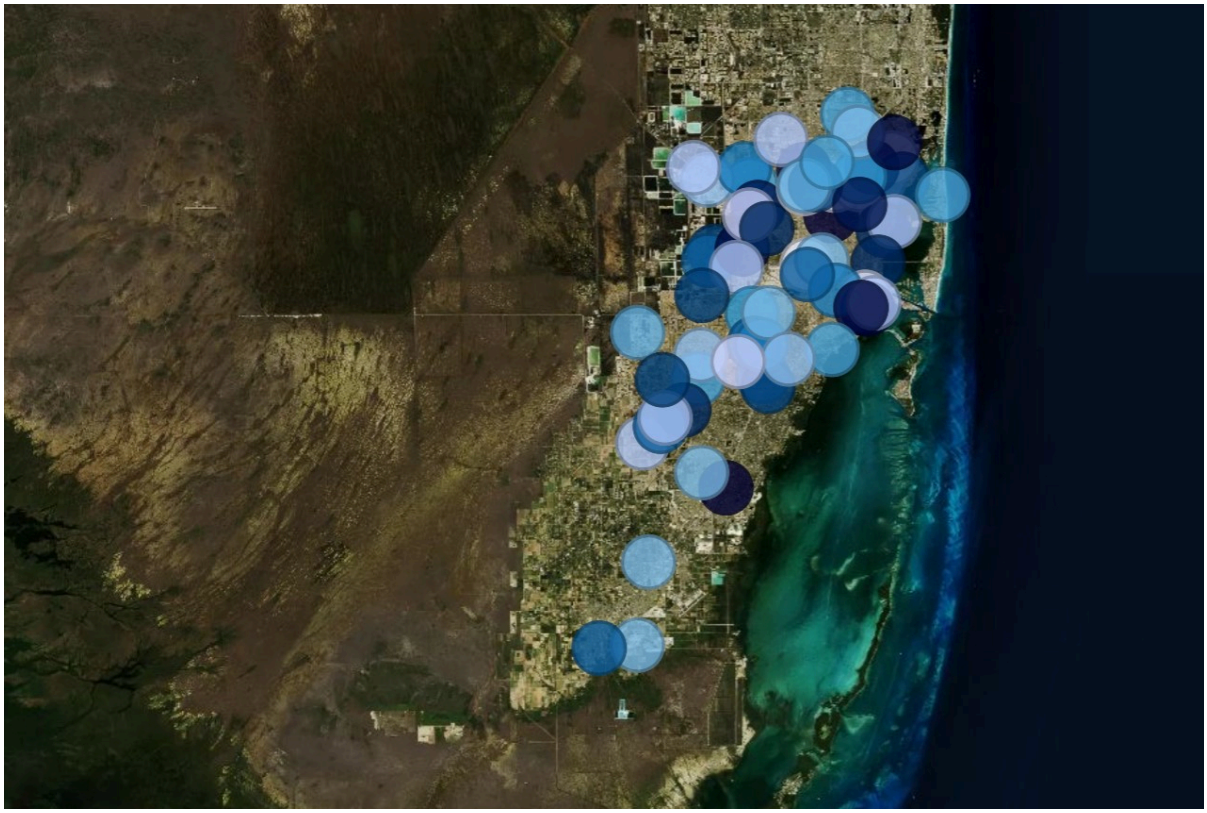
  For more information, see https://www.mapbox.com/help/adjust-tileset-zoom-extent/.
- To ensure that auto-zoom works in IBM Planning Analytics Workspace set the unique polygon identifier as a string value.

- If you work with custom polygons in small areas, for example at postcode level, turning off auto zoom after the initial data loads. This results in a better experience when you filter because the map does not reset zoom and zoom in multiple times.
- Keep the Mapbox keys confidential.

Example of custom regions




Example of custom points



## Marimekko

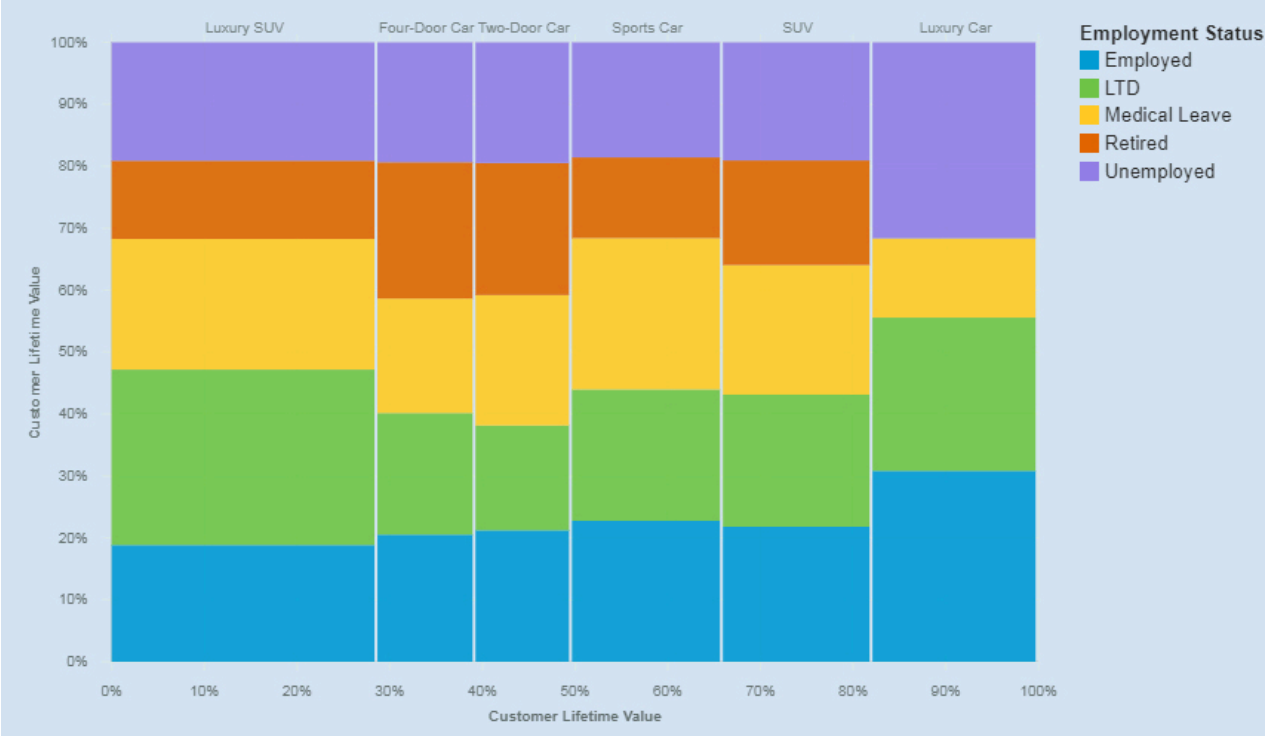
A marimekko visualization is similar to a stacked column visualization. It shows data through varying heights and includes an added dimension of data through varying column widths. The width of the columns is based on the value that is assigned to the width field. Individual segment height is a percentage of the respective column total value.

You can quickly spot large segments, such as a specific vertical that has a large share of a region. You can also identify white space such as an under-represented vertical in a specific region.

The marimekko visualization is useful for part-to-whole comparisons, where you need to show an extra measure/variable.

The marimekko visualization allows data to be depicted along two dimensions simultaneously. For example, market segments are often arrayed along the x-axis, with the width of each column corresponding to the financial value of a segment. You use marimekko visualizations in cases, for example, where you want to show the revenue contribution per product line. Or the gross domestic product per country.

The following example shows the contribution of customer lifetime value and employment status in different vehicle classes.



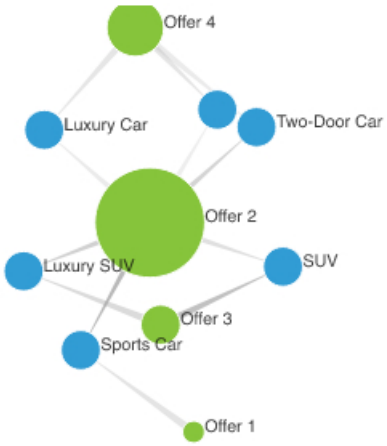

## Network

Use a network visualization when you want to see the connections among columns in your data A network visualization is a good choice to show connections, networks, and points of intersection.

Network visualizations display a set of nodes, represented by symbols, and links, represented by paths, to show the relationship between entities or items.

Use the **From** and **To** fields to define the relationship that you want to investigate.

For example, a network visualization can show offer acceptance by Vehicle Class.

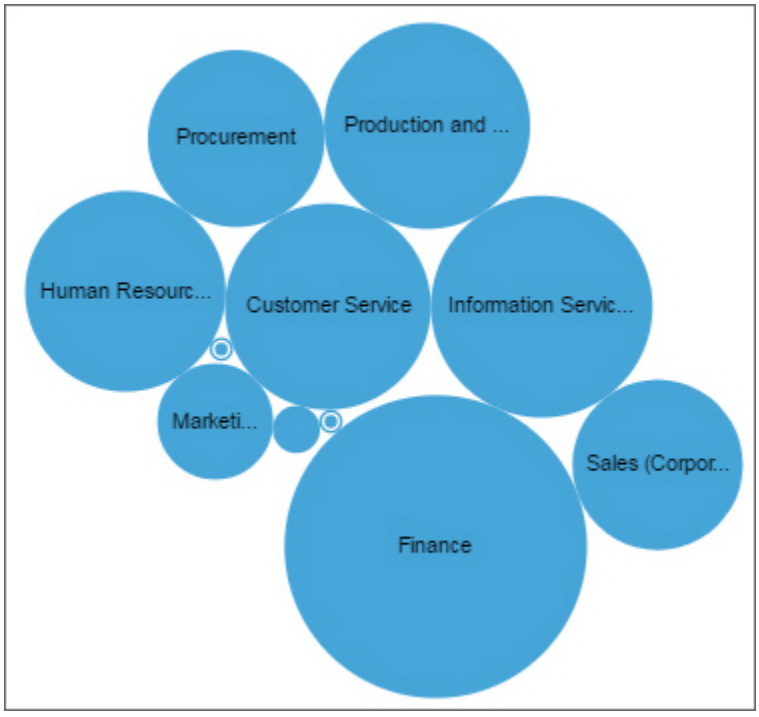



## Packed bubble

Use a packed bubble visualization when you want to show relationships among columns that contain numeric values, such as revenue. It is similar to the bubble visualization but the bubbles are tightly packed instead of spread over a grid. A packed bubble visualization shows a large amount of data in a small space.

The bubbles are in different sizes and colors.

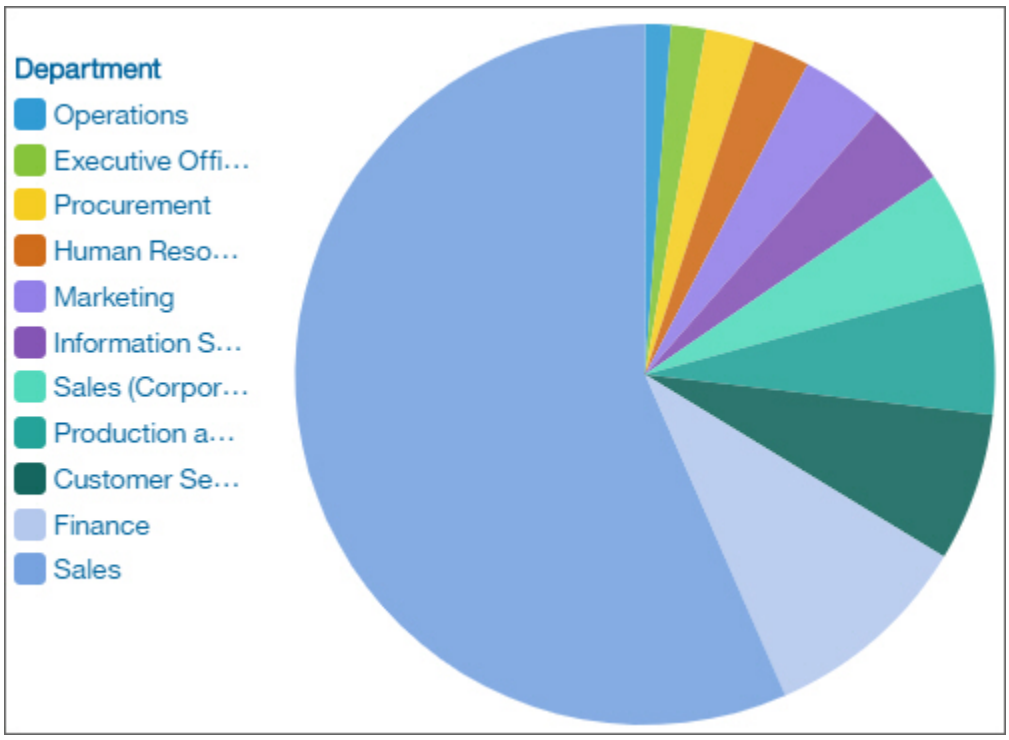
For example, this packed bubble visualization shows external hires by department. Each bubble is a different department. The size of each bubble is determined by the number of external hires for that department.




## Pie

Use a pie visualization to highlight proportions. Each slice shows the relative relationship of each part to the whole.

For example, this pie visualization shows the number of course days for each department.




## Point

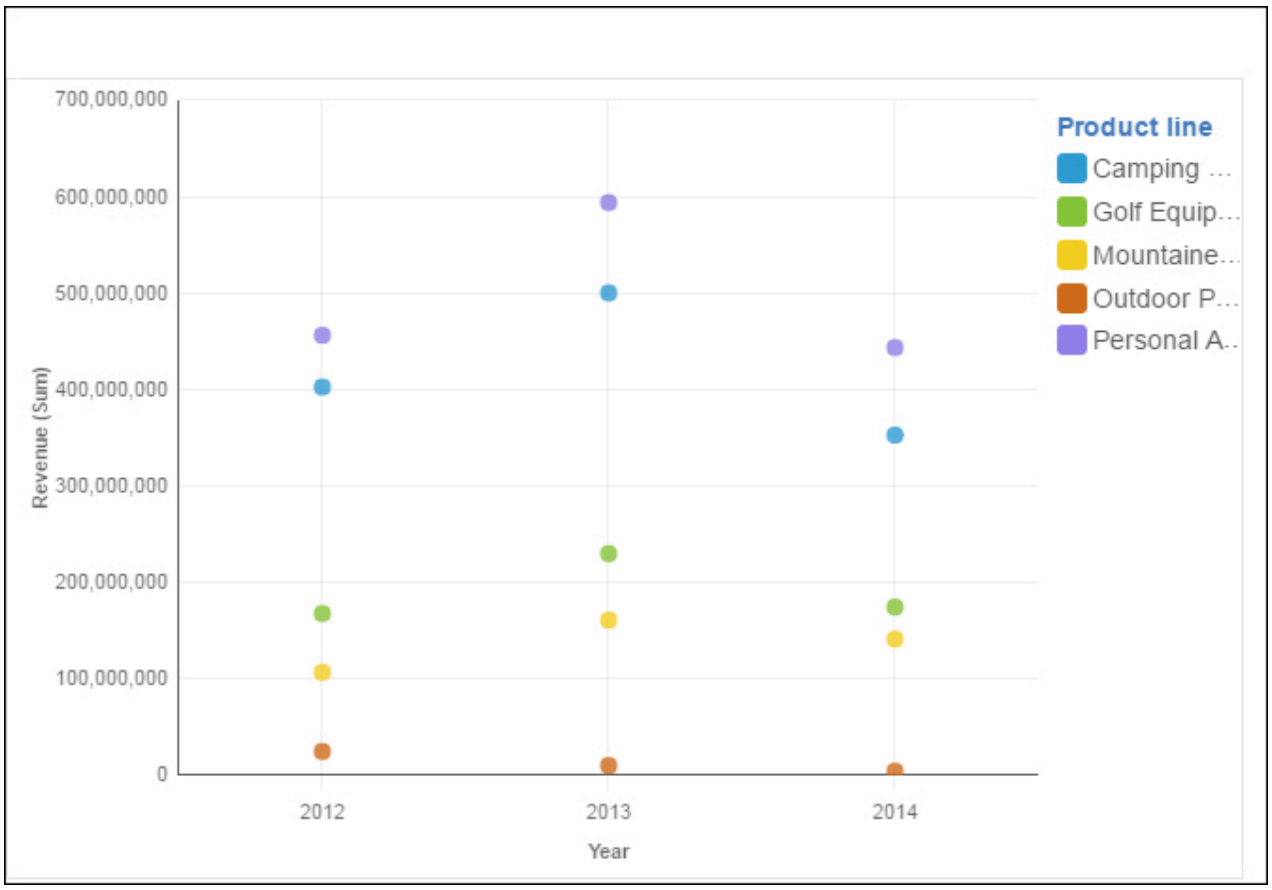
Use a point visualization to show trends over time.

A point visualization can compare trends and cycles, infer relationships between variables, or show how a single variable is performing over time.

A point visualization is like a line chart without the connecting lines.

For an effective line visualization, the x-axis should show time, such as years, quarters, months, or days. If the x-axis shows something else, such as Canada, Netherlands, UK, and US, use a bar visualization.

Data values are plotted vertically.




## Radar

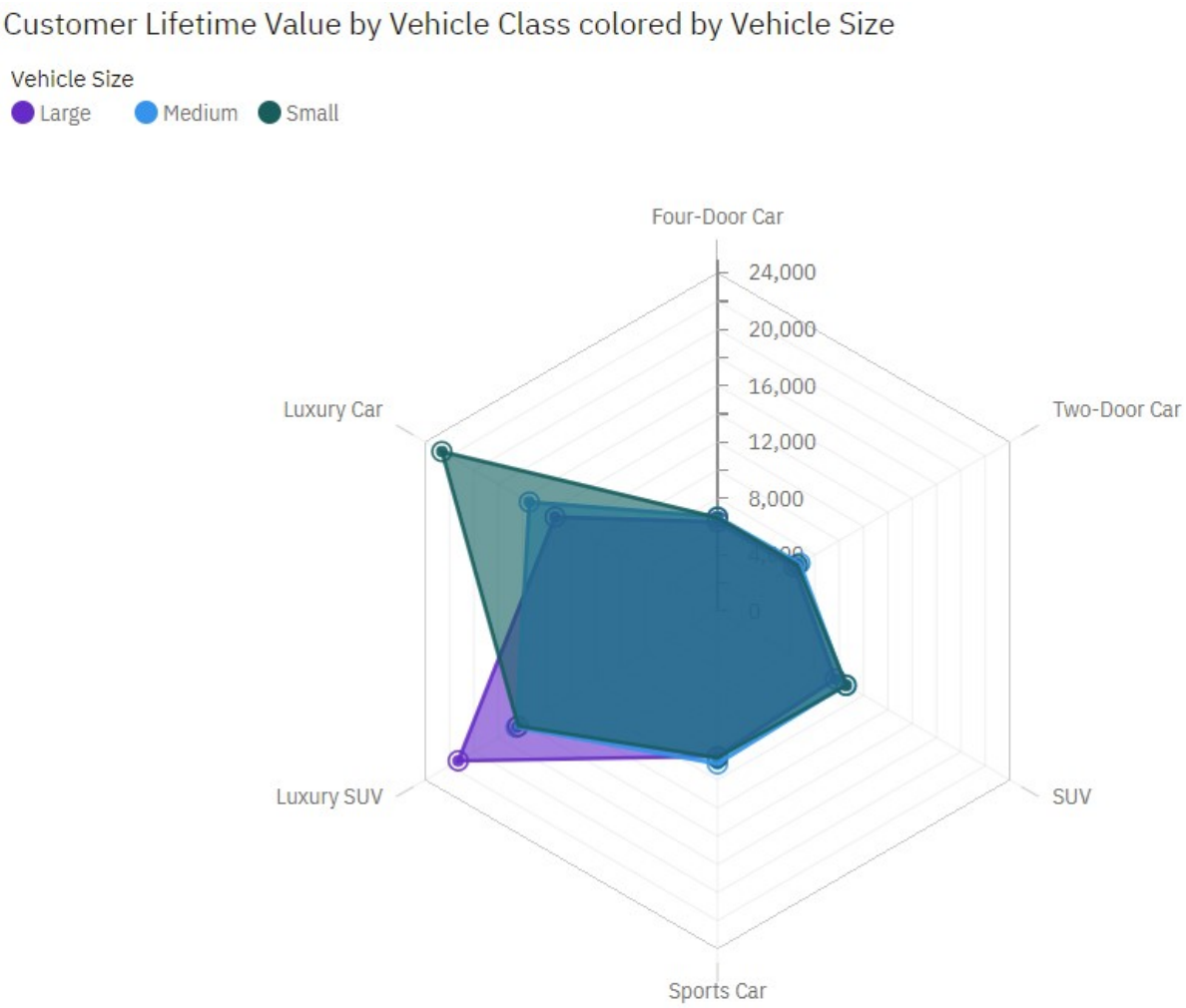
Use a radar visualization for comparing multiple quantitative variables. The radar visualization shows which variables have similar values or if there are any outliers amongst each variable.

Radar visualizations are also useful for seeing which variables are scoring high or low within a data set, making them ideal for displaying performance.

Examples of using a radar visualizations are:

- Comparing cars: speed, durability, comfort, power, space.
- Competitor profiles: number of employees, revenue, profit, current stock price, customer satisfaction.

The following example shows the revenues per retailer type for product lines in different states.

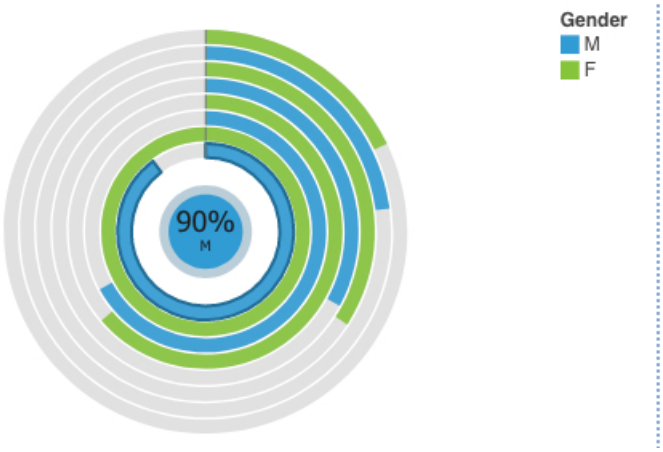



## Radial

In a radial visualization, each bar appears in a circle with longer bars that represent larger values. Hover over a bar to see the details about it, such as the exact value represented by the bar. Each bar starts at 12 noon and goes in a clockwise direction for positive values and counterclockwise for negative values.

Radial visualizations, also known as dial charts or speedometer charts, show information as reading on a dial. The radial visualization is valid only with one category.

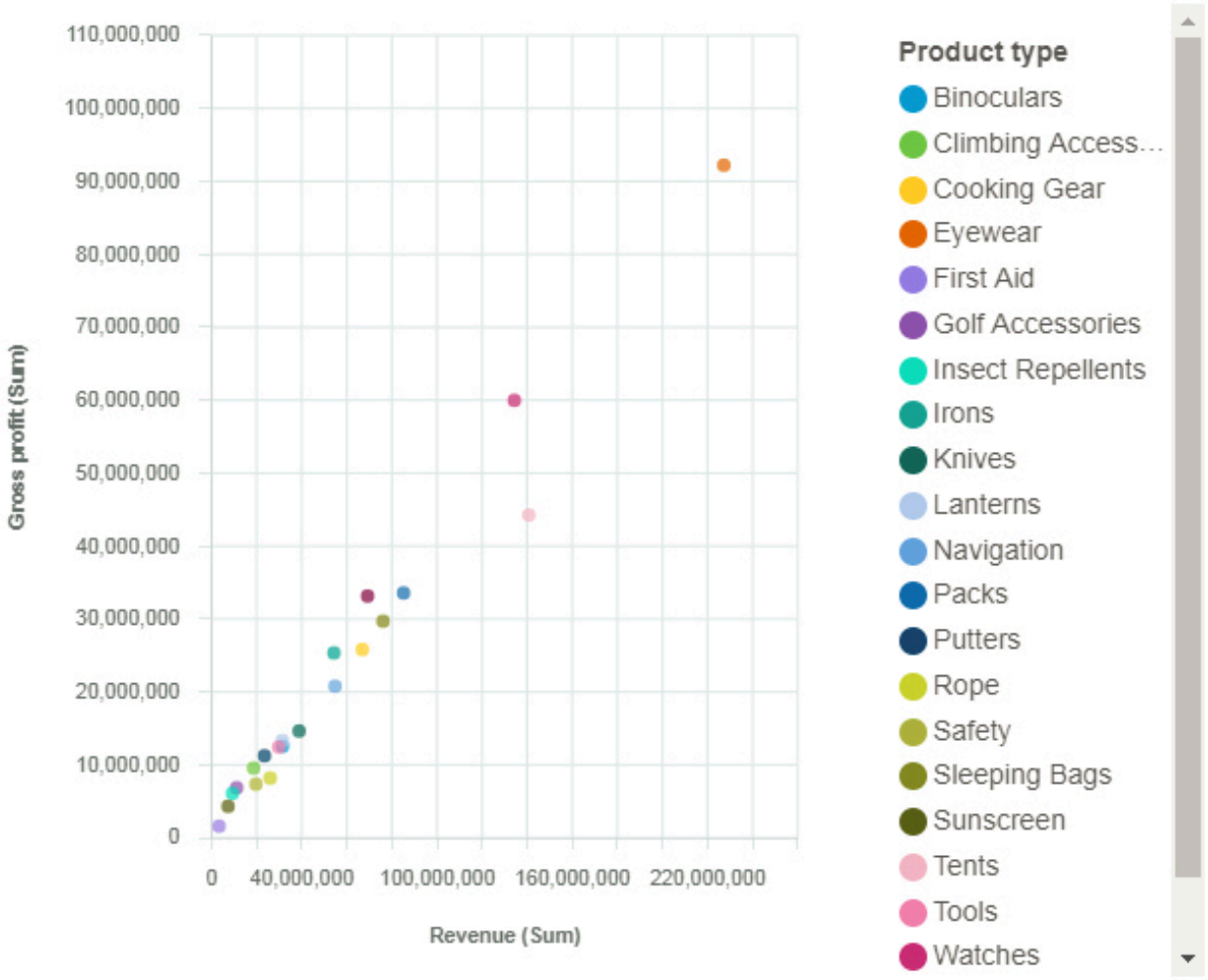
For example, this visualization shows renewals by offer type and gender.



## Scatter

Scatter visualizations use data points to plot two measures anywhere along a scale, not only at regular tick marks.

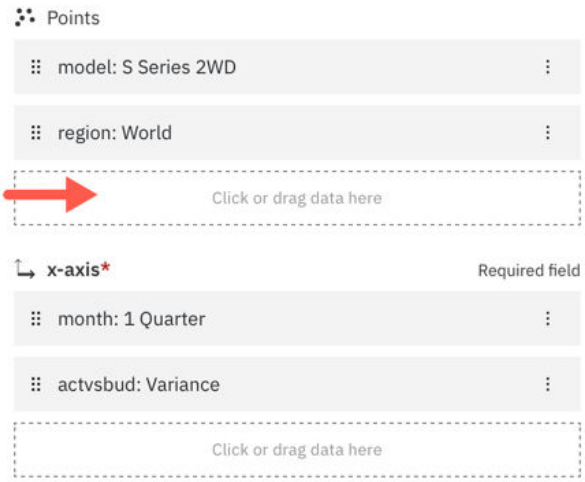
Scatter visualizations are useful for exploring correlations between different sets of data. The following example shows the correlation between revenue and gross profit for each product type.




## Assigning multiple data items to a field

You can assign more than one data item to a field in Scatter visualizations.

To assign a data item to a field, select the data item from the data tree, Filters, or another field. Then, drag the data item to the **Click or drag data here** drop zone under the field you want to map in the **Fields** tab.




When you drag data into an empty field, settings for that data item get mapped to the field. To replace a data item with another one, drag the new data item and place it on top of the existing one.

You can also rearrange the data items under a field by moving them up or down.

## Selector

Use the **Selector** visualization to pick members, sets, or levels for a view (either in exploration or visualization).

The selector widget can have multiple selectors with the same hierarchy that lets you pick members from different groups to form a set and supports one-way synchronization. The multiple selector feature in the selector lets you easily compare members, view data for a range of members, or create a multiple selection list.

When a selector is synchronized with a view, the selection of a member, set, or level in the selector is applied to the view.

For more information on how to create and use selector widgets, see "Create a Selector visualization to manage views" on page 457.

### Adding a selector widget

You can add selector widgets from an exploration. Simply drag and drop the dimension that you want to use to create a Selector, to an empty space in the book. Or, you can follow these steps to create a Selector widget from an exploration row, column, or context area. Keep in mind, you cannot create a Selector from benched dimensions.

1. In Edit mode, click the menu icon next to the dimension you want to create a selector widget from.
2. Click either **Add selector tile** or **Add selector list** and select the tile or list type. You can add the following types of selectors to your book:
   - single tile
   - selector with multiple tiles
   - selector with range selection tile
   - single list
   - selector with multiple lists
   - selector with multiple selection option

**Note:** The range selection tile and multiple selection list selector types can only send synchronization events. They don't receive synchronization events.

You can add a selector from the Data tree. In the Data tree, fully expand the Dimensions node under the cube from which you want to create a selector widget. Click either **Add selector tile** or **Add selector list** and select the tile or list type.

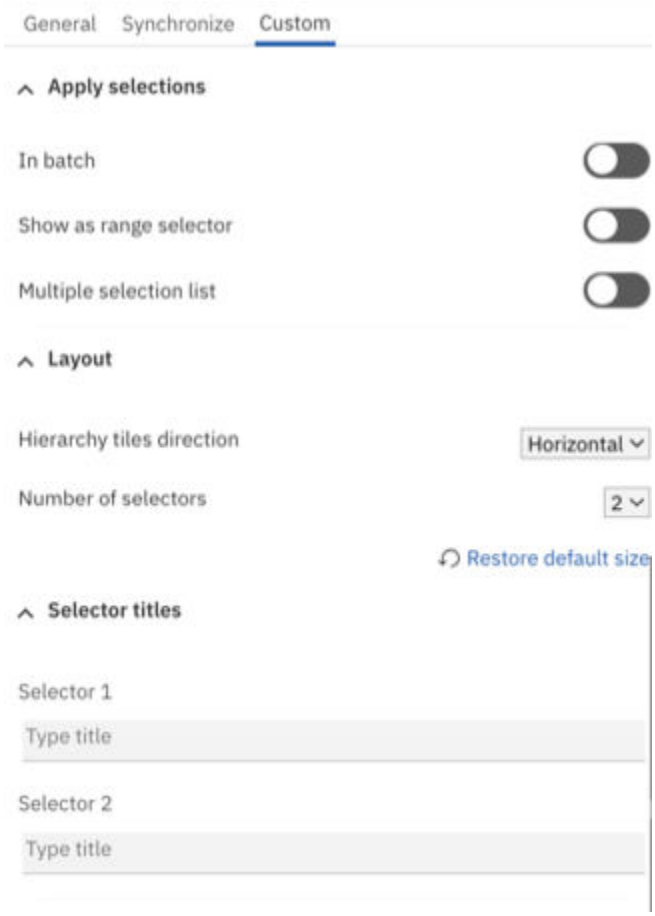
**Note:** You can create a selector only from the Dimensions node under a cube. You can't create a selector from the main Dimensions node.

You can also add a **Selector** to your book from the **Visualizations** pane.

After you add a selector widget to your book, you can customize the widget in its appearance, layout, selection methods, and choose when selection changes apply to the synchronization group.

## Custom properties of a selector widget

You can tailor the Selector widget for your purposes.



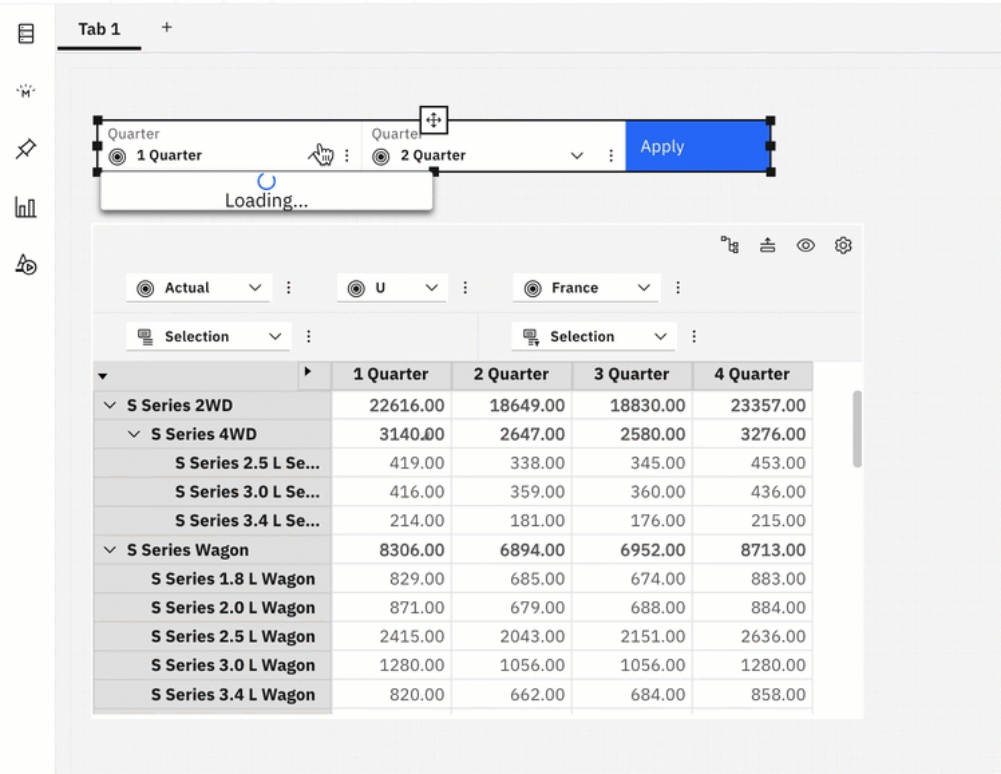

### Apply selections

You can toggle if the change in one selection applies to its synchronization group immediately or changes from multiple selectors apply in a batch. Enable the **In batch** property if you want changes from multiple selectors to apply in a batch and only when the **Apply** button in the widget is pressed. Otherwise, any change in a member selection applies as soon as the change occurs.

Select **Show as range selector** if you want to see a range of members. The two selected members in the selector act as the first and last members of the member list.

**Important:** The **Show as range selector** property works only when you have a maximum of two selectors in a selector widget.

In the following example, both the **In batch** and **Show as range selector** properties are enabled. When you change the member selections to **2 Quarter** and **4 Quarter** in the first and second selectors, the view updates to display all the members between **2 Quarter** and **4 Quarter**.




Enable **Multiple selection list** to display the selector widget as a list, which allows multiple member selections.

**Layout**

In the **Layout** section, you can select whether you want the selector widget to display horizontally or vertically and you can restore the widget to its default size.

You can also select the number of selectors that display in a selector widget. You can choose between two to four selectors. By default, selector widgets contain two selectors.

**Selector titles**

You can specify the title for each selector in a selector widget.

**Selector style**

Much like the custom properties of the selector widget, under the **Selector style** section, you can display a selector widget as a simple list, expand and collapse the widget, show the search bar, and much more.

## Single cell

Use the single cell visualization to highlight the value of a selected cell or to enter data to this cell.

**Important:** Keep in mind, when you enter or delete data in a single cell visualization, the data in the associated cube view is also updated.

If you are using Planning Analytics Workspace 2.0.90 or a later version, you cannot delete data in a single cell widget with the **Delete** key but you can still modify the data or clear the data to return a **0**. You can also delete the single cell widget itself.

You can create a single cell visualization and convert it to a view (exploration) or another visualization and vice versa. When you convert another visualization or view to single cell, the visualization displays data from the first cell.

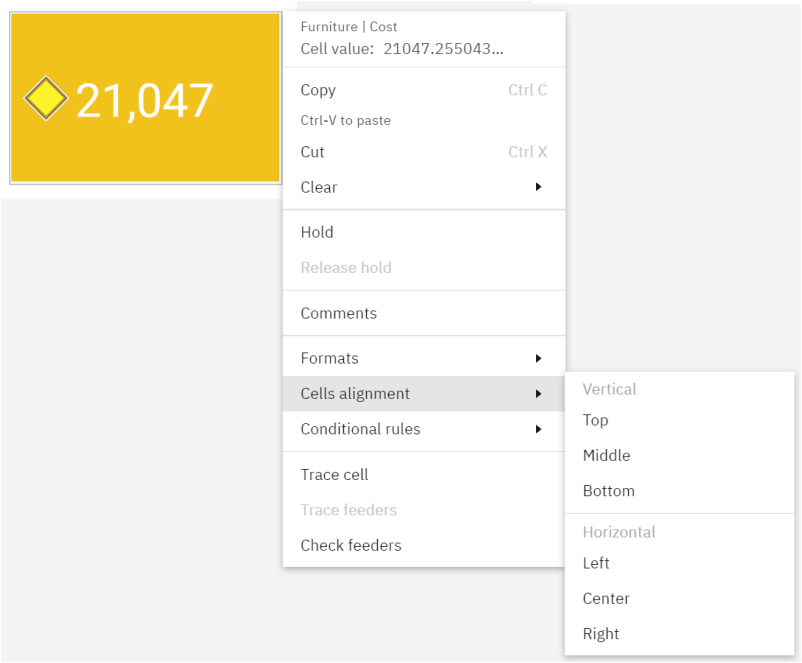
You can add a single cell visualization to your book from **Visualizations** and drag a cube or view on to the template. You can also create a single cell visualization by right-clicking a view and selecting **Add to sheet**.

**Note:** Unless you select a specific cell in the view, when you add a single cell visualization with this method, the visualization displays data from the first cell in the view.

You can change the single cell visualization through the properties under the **Properties** and **Fields** tabs. Select the single cell visualization and go to the **Fields** tab to view and change dimensions and filters. Under the **Properties** tab, you can change the color palette for any conditional rules that are applied to the visualization and enable automatic refresh.

You can change the **Fill color** of **Single Cell** widgets. Select the single cell visualization and go to the **Properties** tab. Under the **General** tab, click **Appearance** to see the **Fill color** options.

You can also change the text alignment of single cell visualization. Right-click the visualization to see cell alignment and other formatting options.

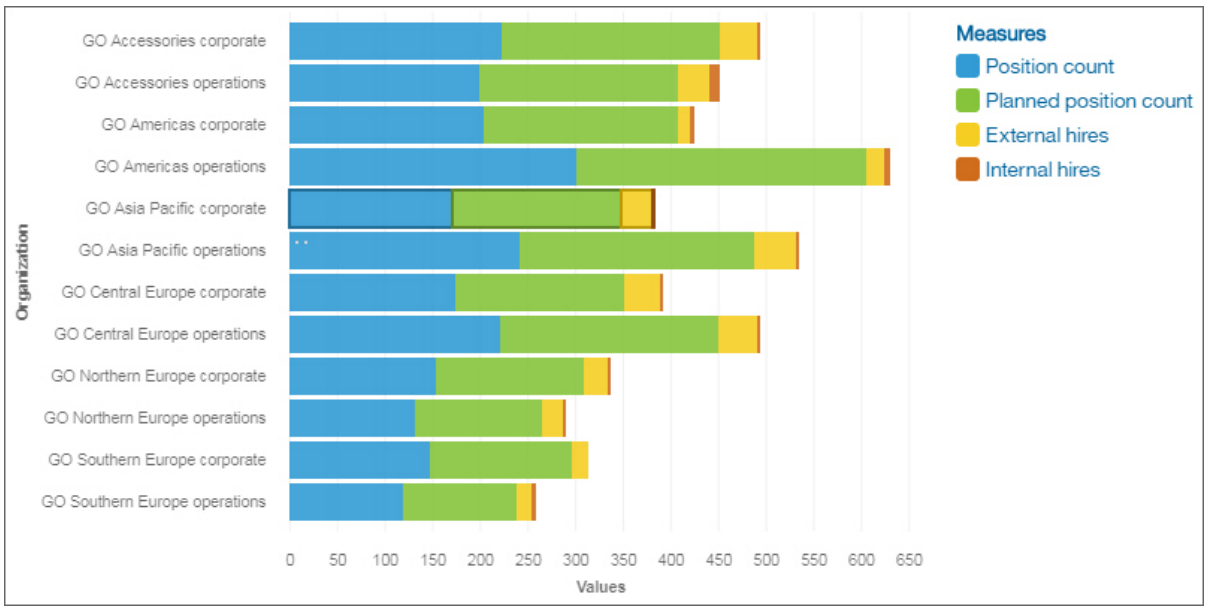



In Planning Analytics Workspace 2.0.92, the **Show as cell** property for single cell widgets was updated to control only the display of an outline around values in single cell widgets. A new property, **Enable editable cell**, allows modification of values in single cell widgets. For more information, see Changes to single cell widget properties.

## Stacked bar

Use a stacked bar visualization to compare the proportional contributions for each item to the total, such as sales for products and sales for products each month.

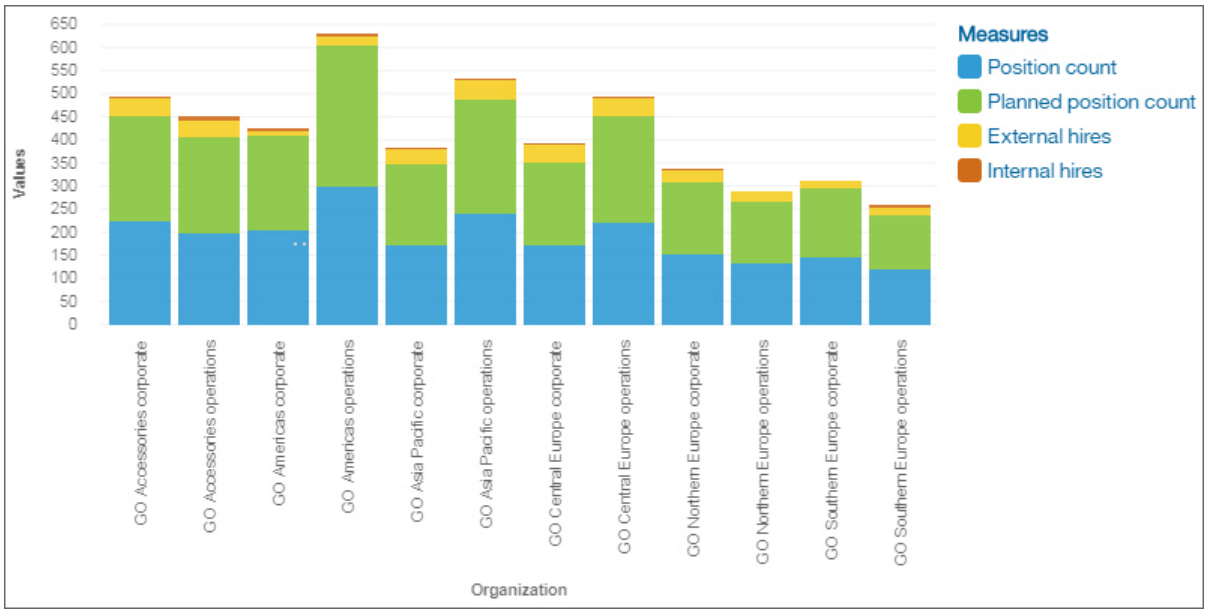
A stacked bar visualization can show change over a specific time period or compare the proportional contributions for each item to the total. If there are so many bars that the labels are impossible to read, filter the data to focus on a subset of the data or use a tree map.




## Stacked column

Use a stacked column visualization to compare the proportional contributions for each item to the total, such as sales for products and sales for products each month.

A stacked column visualization can show change over a specific time period or can compare the proportional contributions for each item to the total. If there are so many bars that the labels are impossible to read, filter the data to focus on a subset of the data or use a tree map.



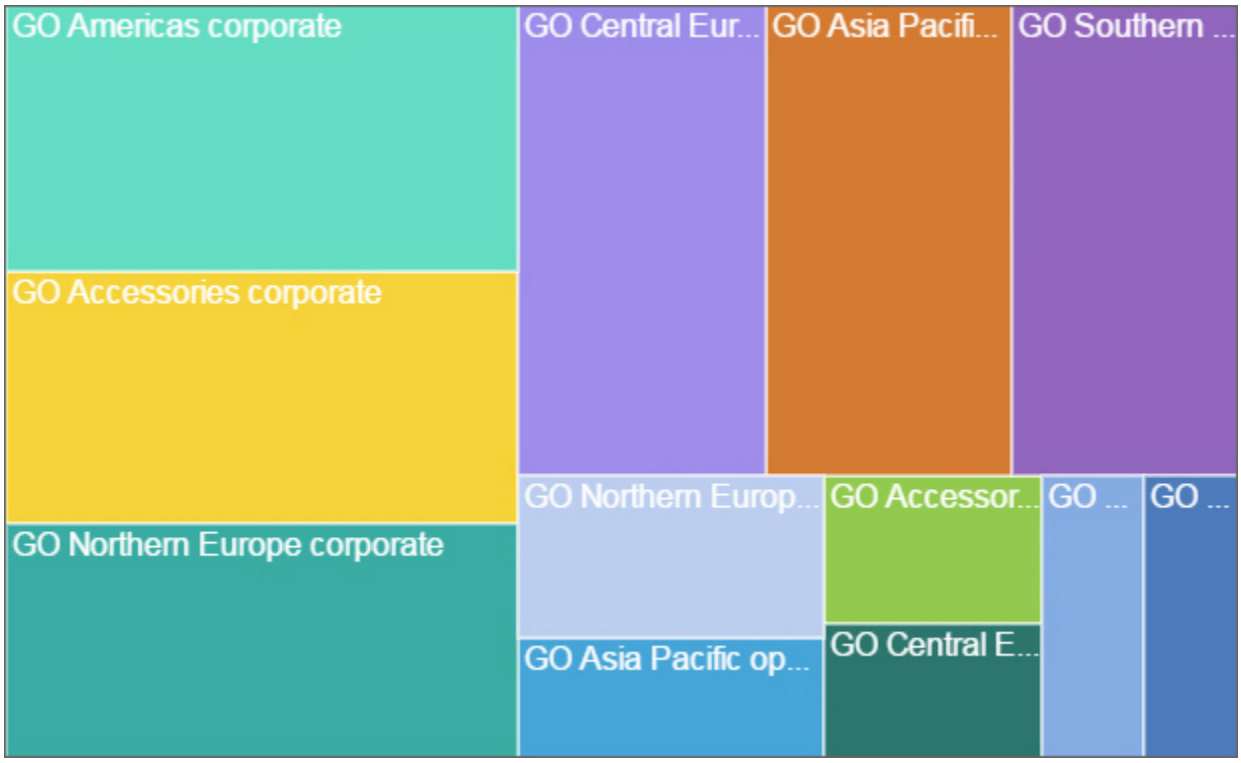

## Treemap

Use a tree map visualization to identify patterns and exceptions in a large, complex data asset.

Treemaps show relationships among large numbers of components by using size and color coding in a set of nested rectangles.

A treemap that is colored by category identifies the level 1 category by color. The sizes of the rectangles represent the values. In a treemap that is colored by value, the sizes of the rectangles represent one of the values and the color represents a second set of values. Do not use data that includes negative numbers. A treemap ignores negative numbers.

For example, this treemap visualization shows course cost by organizational unit.



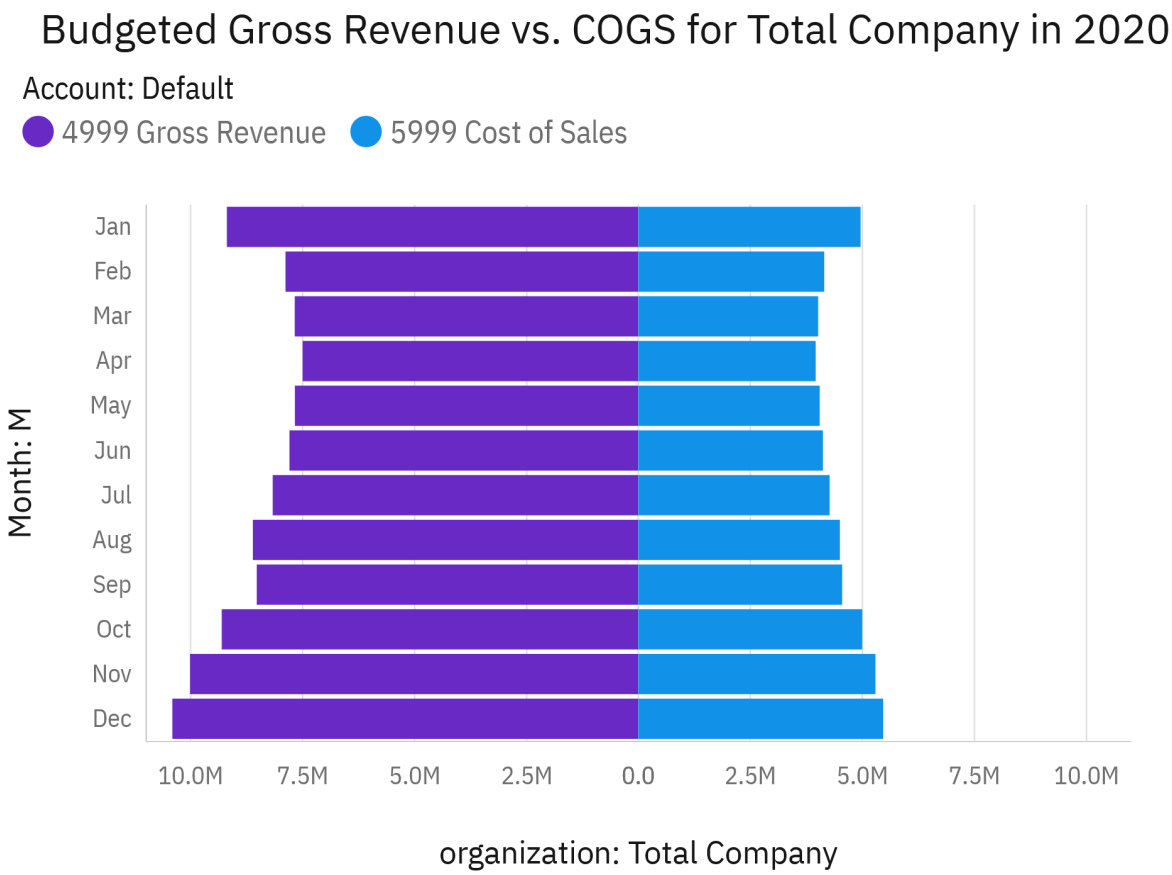

## Tornado

Use a tornado visualization to compare two categories of data series.

The tornado visualization consists of two adjacent bar charts that begin in the center of the value axis and point in opposite directions.

A tornado visualization is similar to the “Stacked bar” on page 512 visualization.

The following image shows an example of a tornado visualization.

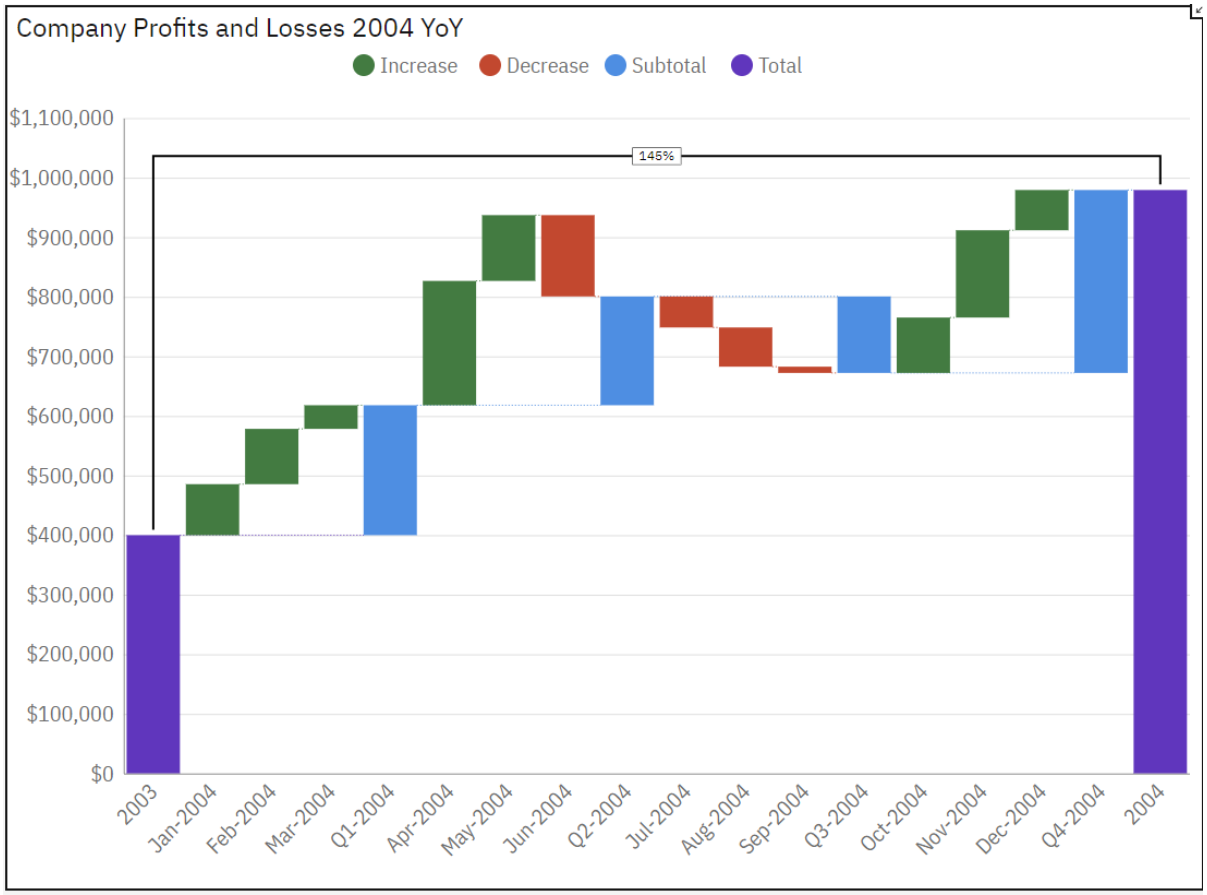



## Waterfall

This visualization is used to show some kind of a measure over a period of a year. The previous year is shown as a starting point. Then the visualization lists the numbers for each month or quarter, and ends with the total for the entire year.

The intermediate values can be either time-based or category-based. Consolidations are treated as subtotals.

The following example shows a typical waterfall visualization:




The following two options in the waterfall visualization significantly improve the user experience:

- The **Change the bar type as** option in the bar right-click menu.

  Use this option to change the bar type to one of the following types: **Auto**, **Delta**, **Adjustment**, or **Subtotal**.

- The **Accumulate totals** property under **Visualizations** > **Data**.

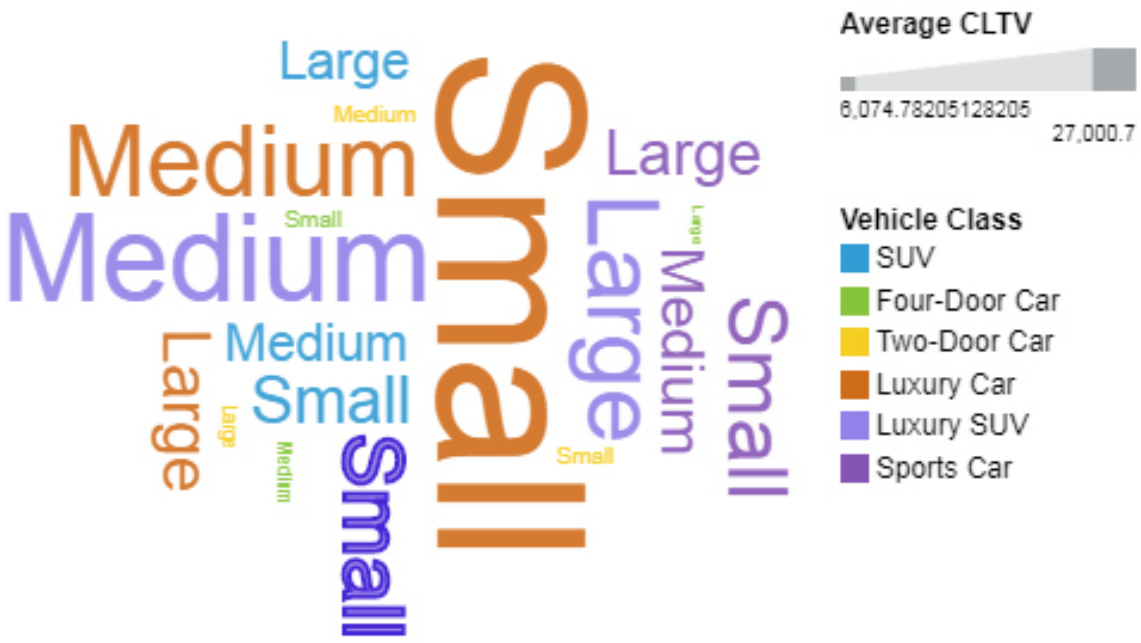
  Use this property to correctly render values from multiple **Total** bars in the visualization instead of changing the TM1 data.

## Word cloud

Use a word cloud visualization when you want to see a text-based visualization of a column. The text height represents the scale. The name itself is the different members of the column.

**Tip:** The view should contain at least 15 columns and at least 100 rows to create an effective word cloud.

For example, this word cloud visualization shows the customer life time value by vehicle size and class.



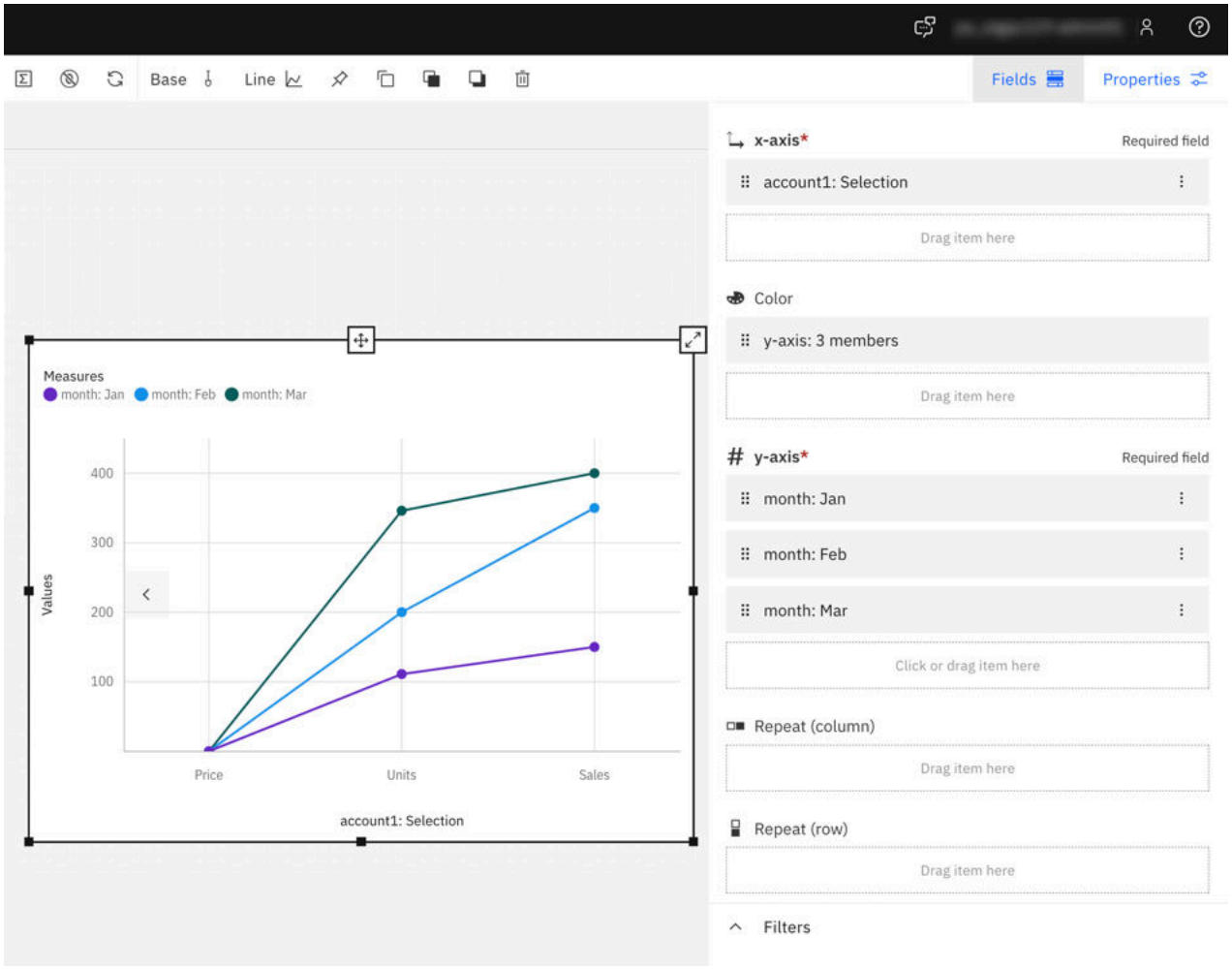

## Plotting members from the same dimension in certain charts

Some visualizations can plot members from the same dimension, which makes it easy to view and compare members quickly.

The following visualizations can plot multiple members from the same dimension:

- Column
- Line
- Line and column
- Bar
- Stacked bar
- Stacked column
- Marimekko
- Point
- Area

In the following example, all members of the *month* dimension display in the **Line** chart as measures on the y-axis.




Notice in the **Fields** panel, the **Color** field shows an item that is called **y-axis: 3 members**, which shows that there are three members that display in different colors on the y-axis.

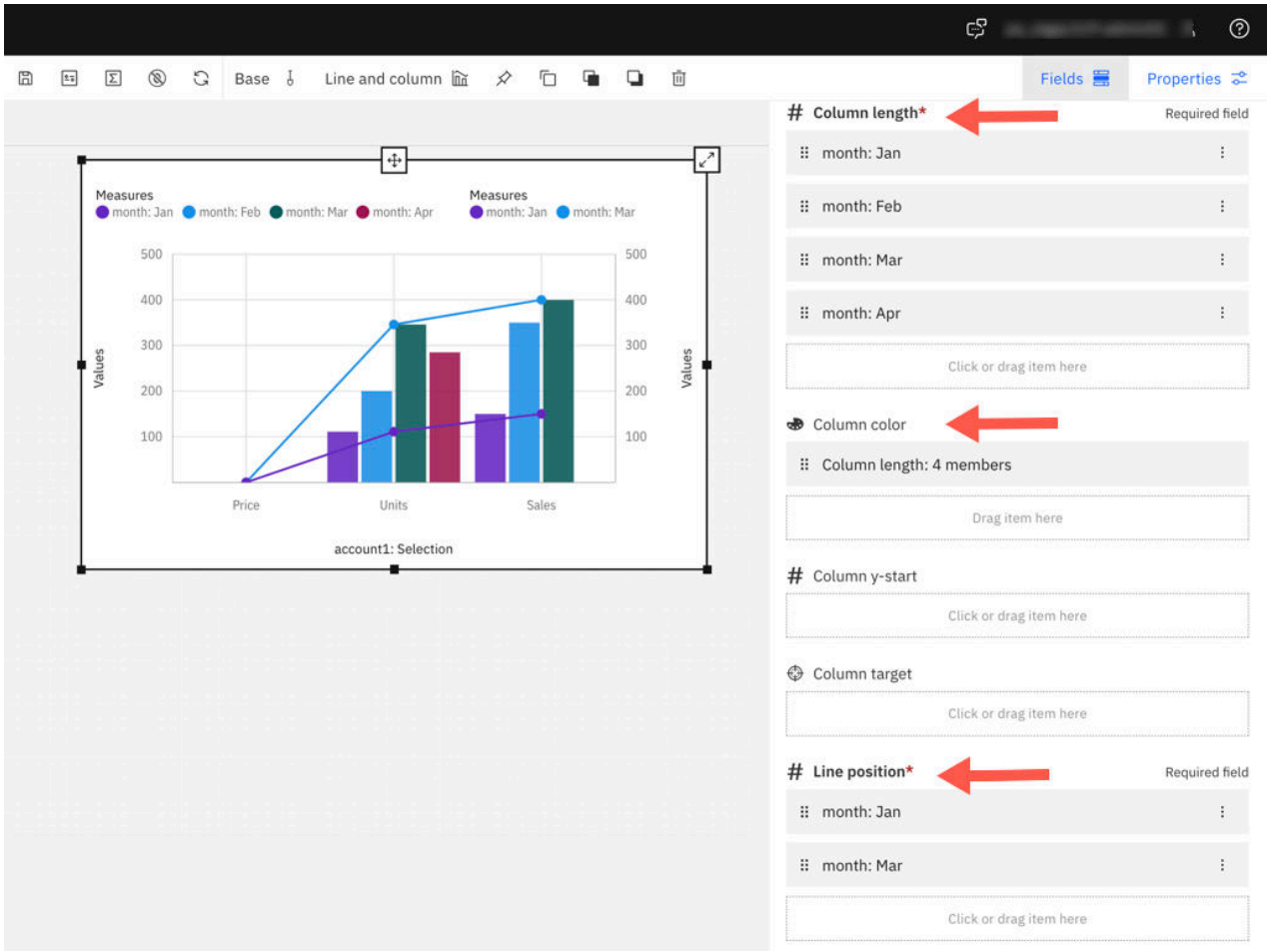
The **y-axis: 3 members** item in the **Color** field acts like any other hierarchy. You can drag it from the **Color** field to other fields like the **Repeat (column)** to see individual charts for each member.




You can click into an empty y-axis field to plot another member. To remove an existing member, click the menu icon for the member you want to remove and select **Remove**. When the set changes, the y-axis field also updates to show the members in the new set.

Similarly, when you create a combination visualization, members of the column dimension are plotted on the **Column length** or **Length** fields and the **Color** field displays the set of members.

In the **Line and column** chart, members of the column dimension appear on both the **Column length** and **Line position** fields. You can then choose which members display as columns and which member should be a line.




## Change the format of data in a visualization

You can set the format of data in a visualization, overriding any existing formatting on an axis. A number of built-in formats are available, or you can specify a custom format.

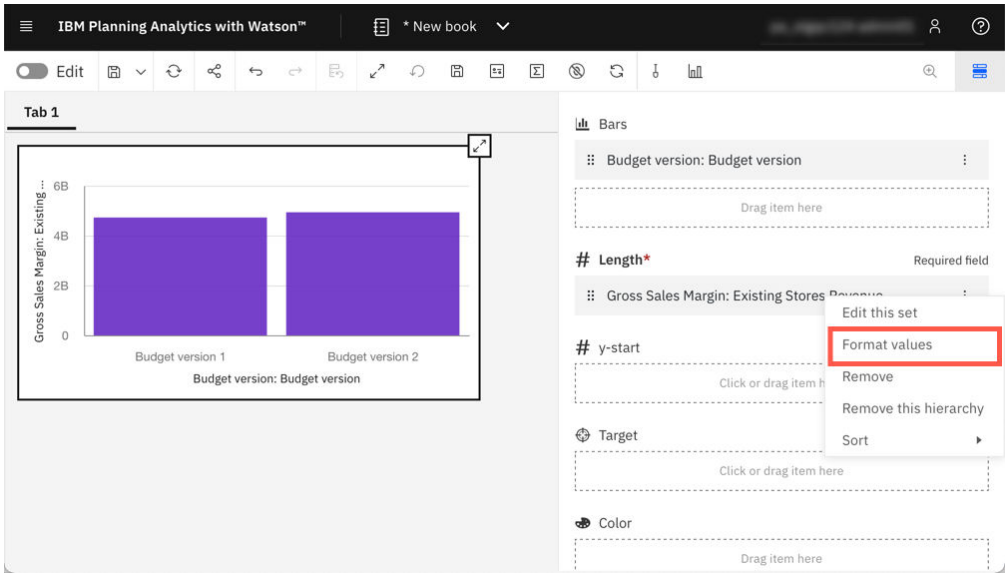
When you set the format on an axis in a visualization, the axis retains the format when you change the member on the axis or when synchronization automatically changes the member.

**Note:** When you set the format on an axis in a visualization, the raw data format is not changed. You can reset the format by selecting **Use database format**.

The formats are also retained when you switch from one visualization to another or convert the visualization to an exploration and vice versa.

To apply formatting to a visualization axis:

1. Select the visualization and click **Fields**.
2. Click the menu icon for the axis that you want to format and select **Format values**.




3. Choose the format that you want to apply to the axis.

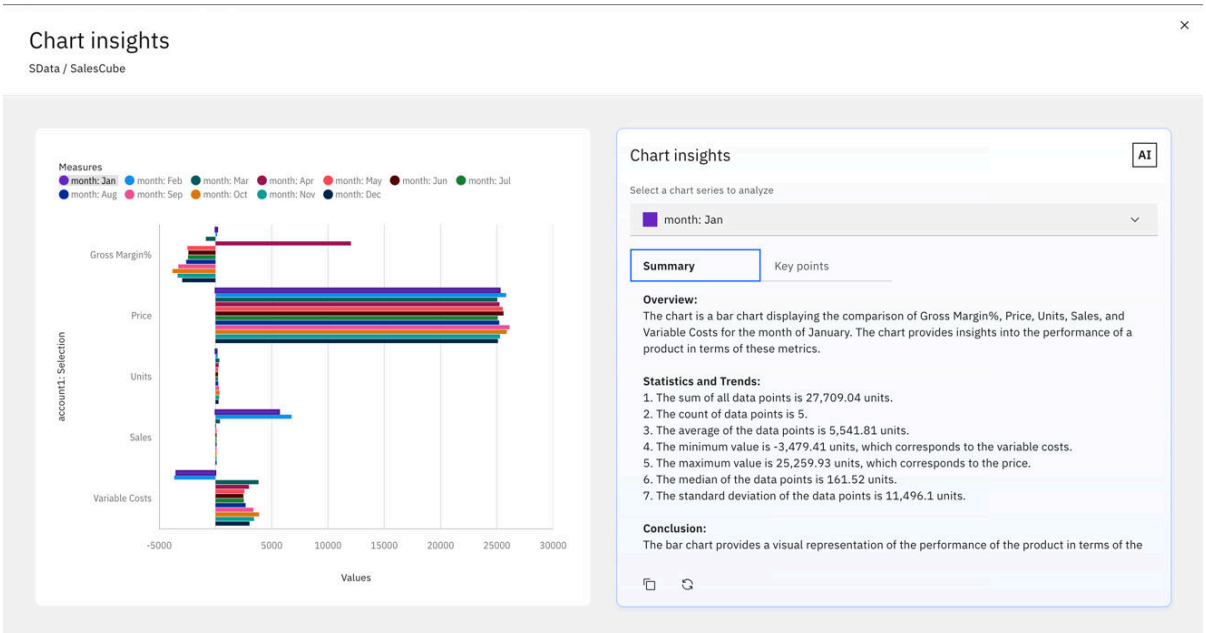
You can also right-click the axis title in a visualization and click **Format values** to select and apply a format to the axis.

## Use AI-generated insights to interpret a visualization

**Chart insights** provide AI-generated insights in the form of text summaries and key points to provide an easier interpretation of data in a visualization. The initial release of the **Chart insights** feature supports English only and is available only in Planning Analytics Workspace Cloud.

**Important:** The **Chart insights** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

With **Chart insights**, depending on the visualization type and the available data, you can review important information that helps you interpret a visualization. The **Summary** tab variably provides an overview, identification of statistics and trends, conclusion, and analysis relevant to the contextual information in the chart. The **Key points** tab provides a summary of the visualization, trend conclusion, brief listings of the key statistics, notable observations, and insights.

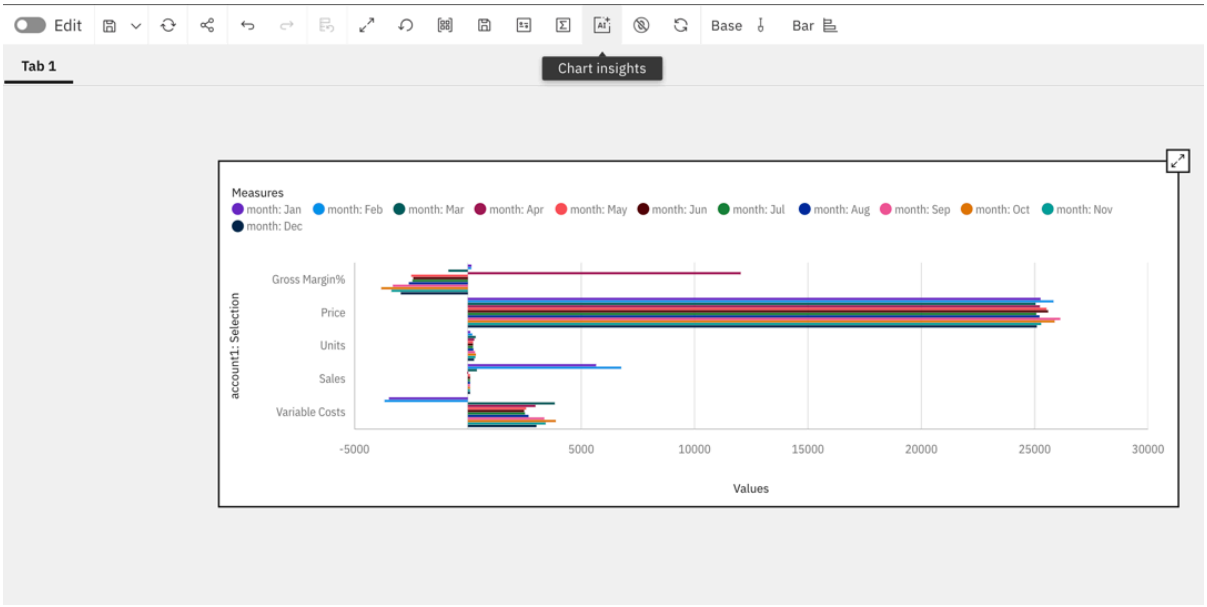


You can view details of the AI model and an overview of how it works in this feature by clicking the **AI** label.

## Generating insights

To generate AI insights for a visualization:

1. Select the visualization that you want the analysis for and click **Chart insights** on the toolbar.

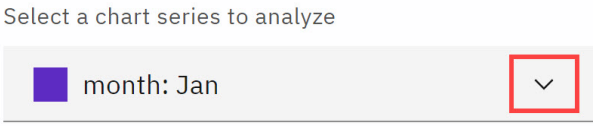



   AI starts generating a text summary of the data in the selected series.

2. After the summary is complete, you can click the **Key points** tab for insights and notable observations.

   **Note:** You cannot navigate between the **Summary** and **Key points** tabs during text generation. You also cannot regenerate or copy the response until after generation is complete.

3. If your visualization has more than one series, AI insights are generated for the first series in the visualization. For example, if your visualization includes a series that includes individual months, the insight is generated for January. To generate insights for a different series, click the **Select a chart series to analyze** menu and select a different series.




4. Optionally, click **Regenerate response** to regenerate the summary and key points. The response will vary slightly each time you regenerate.

**Chart insights** supports the following charts for analysis:

- Area
- Bar
- Box plot
- Dual axes column
- Dual axes line
- Line
- Line and column
- Pie
- Point

- Radar
- Radial
- Stacked bar
- Stacked column
- Waterfall
- Word cloud

### AI explained

The Chart insights feature uses granite-34b-code-instruct from the Granite family of IBM foundation models to generate summaries.

The Granite models are decoder-only models that can efficiently predict and generate language. These models were built with trusted data that has the following characteristics:

- Sourced from quality data sets in domains such as finance (SEC Filings), law (Free Law), technology (Stack Exchange), science (arXiv, DeepMind Mathematics), literature (Project Gutenberg (PG-19)), and more.
- Compliant with rigorous IBM data clearance and governance standards.
- Scrubbed of hate, abuse, and profanity, data duplication, and blocklisted URLs, among other things.

**Note:** IBM is committed to building AI that is open, trusted, targeted, and empowering. For more information about contractual protections that are related to IBM indemnification, see the IBM Client Relationship Agreement and IBM watsonx.ai service description.

To generate a summary and insights, the AI model takes key inputs such as the chart data and calculations from the database. The AI model can analyze information from the selected chart series and calculated statistics to describe the chart, data points, and any trends presented in the data. A text-based summary is then generated by the model along with key insights for the selected chart series.

### Tips and best practices

While working with Chart insights, here are few things to ensure that might lead the AI model to generate better insights:

- Chart has data and is not empty
- Chart data is in context of the cube
- Cube, dimension, and member names are descriptive and accurately represent the data
- There are no named outliers for dimensions that might confuse the AI model

## Matching color palettes on multiple visualizations

When multiple items on a book share a category or value, you can use the **Color consistency** setting to automatically adjust the visualization colors to match each other.

### About this task

For new books, the **Color consistency** setting is on by default. For upgraded books, the **Color consistency** setting is off by default because you might have selected specific palettes for different visualizations.

Complete the following steps to enable or disable the setting:

### Procedure

1. Click the **Properties** icon .

   If you don't see the **Properties** icon, enable the **Edit** option  for the book.

2. For the entire book, ensure that nothing else has focus by clicking the background, away from any visualization or object.
3. Click the **General** tab, then click **Advanced**.
4. Click the **Color consistency** toggle to turn the setting on or off.

# Setting visualization properties

You can set **Visualization**, **General**, **Synchronize**, and **Custom** properties to determine the appearance of a visualization and its behaviors. For example, you can pick the color palette or set the position of the legend in a visualization. The properties that you can set vary according to the type of visualization you are using; not all properties are available for all visualizations.

## Procedure

1. In **Edit** mode, click anywhere on a visualization.
2. Click **Properties**.
3. Under **Visualization**, set properties for your visualization. Some examples are:
   - **Color** - Pick the color palette that you want to see in the visualization.
   - **Axis** - Under this section, you can choose how the axes display. You can choose to show or hide the item axis or column axis lines, show axis titles, show axis title labels, format the axis titles and axis title labels, and more.

     If the visualization uses repeat column or row filters, you can format repeat column and repeat row titles.

     You can also hide the hierarchy name in an axis title to shorten the label. The axis title shows both the hierarchy name and the name of the set or member that was selected. When the hierarchy name is hidden, you see the name of only the set or the selected member.
   - **Legend** - In visualizations with legends, click **Show legend** to display the legend. You can also select a legend position and set properties for the legend and legend title.

     As in axis titles, you can hide the hierarchy name in a legend title to shorten the label. Hiding the hierarchy name displays only the set or the selected member of the dimension.
   - **Chart** - For eligible visualizations, you can add a zoom bar, hide grid lines, set item labels or value labels to display, improve the visibility of labels, and enable the **Responsive** property to hide certain aspects of the visualization if the size is limited and maximize the space that is available to display data. Enable **Value axis show zero origin** if you want to see the zero origin on the value axis.

     You can also set how data points interpolate in certain visualizations with **Interpolation**.

     For **Tree map** visualizations, enable **Show leaf labels** to see labels for leaf members in the visualization.

     **Line and Column** visualizations have a **Synchronize axes** property. By default, **Line and Column** visualizations use separate vertical axis value ranges for the two measures in a visualization. When **Synchronize axes** is enabled, the visualization uses a common vertical axis range for both measures, resulting in a more accurate visual representation of your data. In Planning Analytics Workspace Classic, this property was known as Share vertical axis range .
4. Under the **General** tab, set general appearance properties for the visualization:
   - Visualization appearance properties, such as **Fill color**, **Border color**, and **Widget opacity**.
   - PDF print settings to specify how you want the PDF file to appear: **In view** to print the visualization exactly as it is shown in the dashboard, or **New page** to display all visualization data in the PDF file.

     If you select **New page**, the PDF file prints all the data from the visualization, regardless of what is shown in the dashboard. The PDF file first shows all the columns in the visualization, then all the rows. The PDF file has a new page after it prints a number of columns up to the specified number, then a new page after it prints a number of rows up to the specified number. You can also specify whether want to fit all columns in one page and all rows in one page.

- Layout properties like alignment, positioning, and size.

5. Set the level of synchronization you want for the visualization under **Synchronize**.
6. Under the **Custom** tab, set the **Set editor** access and **Toolbar** properties.

**Set editor**

Enable the **Allow access to Set editor** option to allow users to access the Set editor from the view. Disable the option to prevent access to the Set editor. When you disable access to the Set editor, users cannot open the Set editor, but they can select from members of the current set.

**Toolbar**

To manage which icons are available on the toolbar for you and your users, you can apply the **Toolbar** icon settings in the **Properties** panel. Click the **Custom** properties tab to view all the available **Toolbar** icons that you can set.

Depending on the visualization, you can choose to display or hide the following available toolbar icons:

- **Save**
- **Format values**
- **Sandbox selection**
- **Change visualization**

When you select both a cube viewer and other visualizations in a book only the common toolbar icons are available.

# Customize sets in dimension lists

Use the **Customize sets** option in a new experience view (exploration or visualization) to select the sets that display in a dimension list. You can select up to 20 sets for a dimension.

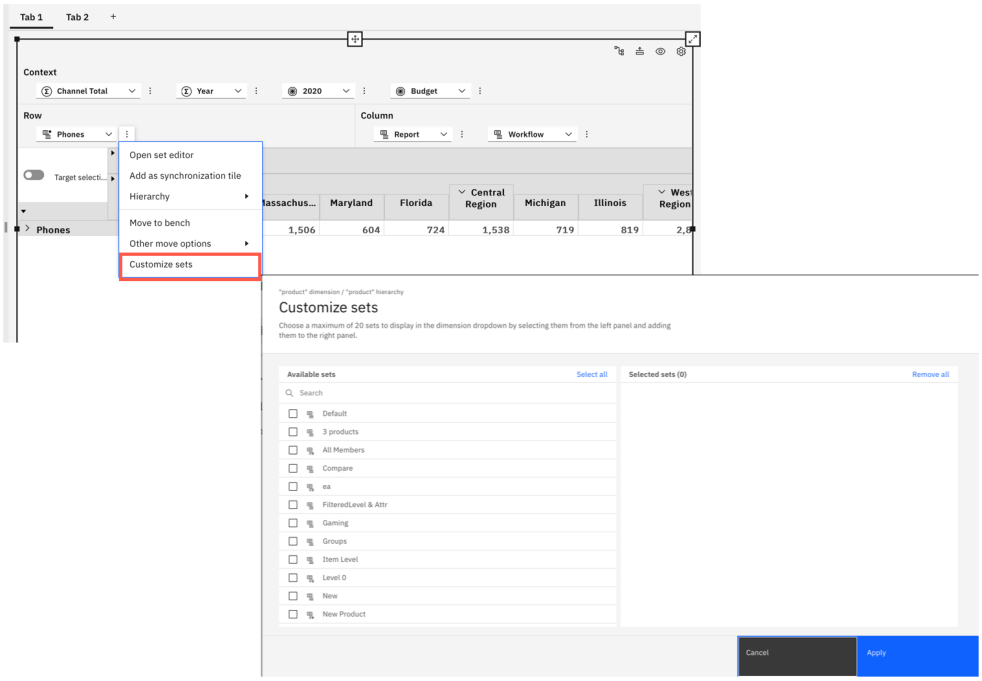
**Note:** The **Customize sets** option does not limit the sets that you see in the set editor. You can still see the full list of sets in the set editor.

## Customizing sets for dimensions in explorations

To select the sets that display in a dimension list in an exploration:

1. Click the menu next to a dimension in the **Row** or **Column** area.

   **Note:** You cannot customize sets for dimensions that are in the **Context** area.




2. In the **Customize sets** window, select the sets that you want in the dimension list from the **Available sets** pane. As you select the sets, they appear under the **Selected sets** pane.

   You can also search for sets and select them one by one or click **Select all**.
3. Click **Apply**.

Now, when you click on the dimension in the view, you see only the sets you selected.

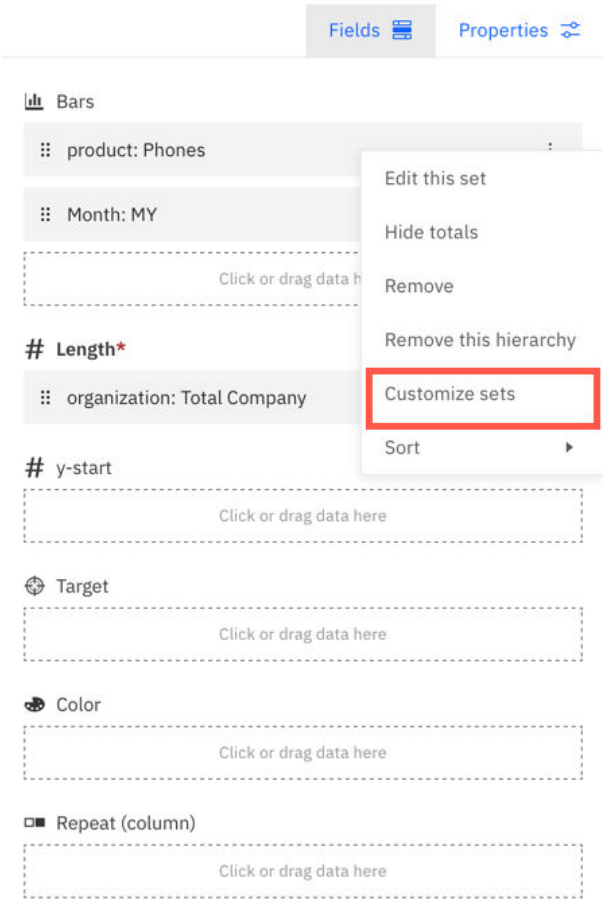
You can move dimensions with customized sets between the **Row**, **Column**, and **Context** areas without losing the limited sets.

However, if you move a dimension with customized sets off the exploration and add it to the book as a widget, the limited list of sets is lost. You will see all the sets for that dimension in its drop-down.

## Customizing sets for dimensions in visualizations

You can also customize sets for dimensions in visualizations:

1. Click anywhere on the visualization in **Edit** mode and click **Fields**.
2. Click the menu next to the dimension that you want to customize and select **Customize sets**.




3. In the **Customize sets** window, select the sets that you want in the dimension list from the **Available sets** pane. As you select the sets, they appear under the **Selected sets** pane.

   You can also search for sets and select them one by one or click **Select all**.
4. Click **Apply**.

Now, when you click on the dimension in the field slot under **Fields**, you see only the sets you selected.

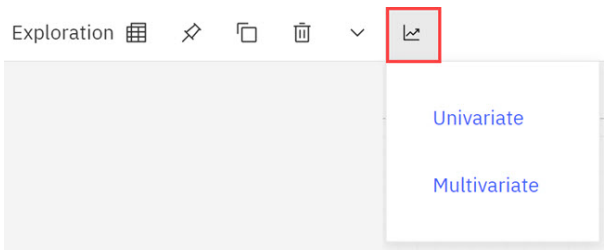
You can switch between a visualization and an exploration (or vice versa) without losing any customized sets that you applied.

# Forecasting

Use forecasting in IBM Planning Analytics Workspace to discover and model trend, seasonality, and time dependence in data.

You can forecast in Planning Analytics Workspace by using automated tools that model time-dependent data. Automated model selection and tuning makes forecasting easy to use, even if you are not familiar with time series modeling.

To begin forecasting, click an exploration view in a book, then click **Forecast** and click either **Univariate** or **Multivariate**, depending on which type of forecast you want to generate.



Forecasts and their confidence bounds (when applicable) are displayed in visualizations as a continuation of historic data. You can also view the statistical details for generated models if you want to see the technical background.

Specifying time series in forecasts often requires data manipulation. Planning Analytics Workspace supports a wide range of time series without the need for manipulation, ranging from standard date and time types, to nested periodic and cyclical time fields. When data is recognized as a time series, data preparation is automated. Appropriate trend and seasonal periods are detected.

You can view forecasts in line visualizations. Forecasts and confidence bounds are computed for each time series and displayed in the visualization as extensions of the current data. You can inspect each time series separately and tailor the forecast and results to your own data and requirements.

If you are familiar with forecasting models, you can view the selected model type, estimated model parameters, standard accuracy measures, and processing summary information when you preview a forecast.

When the forecast options are set up, you can preview the forecast. You can also export the forecast values into a CSV file.

## Univariate and multivariate forecasts

You can create both univariate and multivariate forecasts in Planning Analytics Workspace

### Univariate forecasts

A univariate forecast predicts the future values of a single variable based on historical data, using the Holt-Winters model.

### Multivariate forecasts

A multivariate forecast takes into consideration multiple time-dependent variables. The forecast model looks not only at historical data, but also the dependencies between the specified variables to create a forecast. The selected variables play a significant role in driving the forecast results. Therefore, it is important to select variables that have high correlation with the historical data.

Multivariate forecasting uses one of two models, vector auto-regressive (VAR) and auto-regressive integrated moving average (ARIMAX). If the historical data consists of less than 64 data points, VAR is used. If future values are included in the forecast, ARIMAX is used. If the historical data consists of more than 63 data points, but future values are *not* included in the forecast, then the optimal model is used. You have the option to include or exclude future values when you configure your forecast.

The variables for a multivariate forecast can be selected from different cubes on the same Planning Analytics database, as long as both the historical cube and the variable cube use the same time dimensions. Time dimensions are the dimensions placed in the columns position of a view to represent a continuous set of data points.

Note that neither multivariate models are seasonal. If your data is highly seasonal, then univariate forecasting is recommended.

## Forecasting requirements

To use univariate or multivariate forecasts, the time dimension, historical data, forecast data, and version dimension (if you want to use one) must meet certain requirements. For multivariate forecasts, there are also requirements for forecast variables.

Some requirements vary between univariate and multivariate forecasts, as noted in the following sections.

## Time dimension

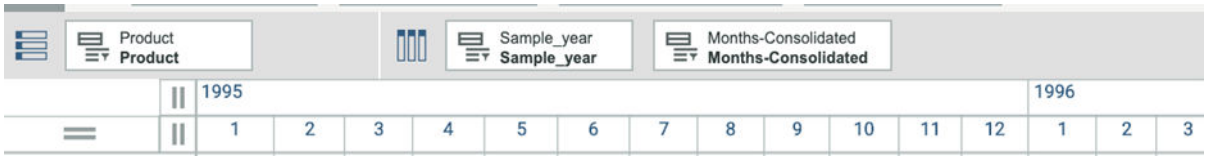
Forecasting requires the time dimension to be placed on the column. The algorithm requires the time dimension members to be organized in order of increasing time, where each member of a level represents a consistent unit of time. Members within the dimension definition that do not conform to this requirement, for example calculated year-to-date member, can be hidden. Relevant time periods can be chosen to be ignored, resulting in a linear interpolation of values to be predicted from. Forecasting also supports hierarchical time dimensions and nested dimensions as time: year, quarter, and month dimensions. Views with dimensions that do not represent time on the column can be forecasted, but the process might fail or the results might be unreliable.

**Note:** When the members of the dimensions used to indicate the time (columns) have multiple parents, then the forecasting fails. With multiple parents, the correct order of those members is not known. To solve this limitation, create an alternative hierarchy for the dimensions that exclude the extra parents.

One or more time dimensions can be used to forecast against, and they must all appear on the column of the view. In either case, the leaves must result in contiguous, evenly spaced periods of time. The following screen capture displays an example of a single time dimension of years:



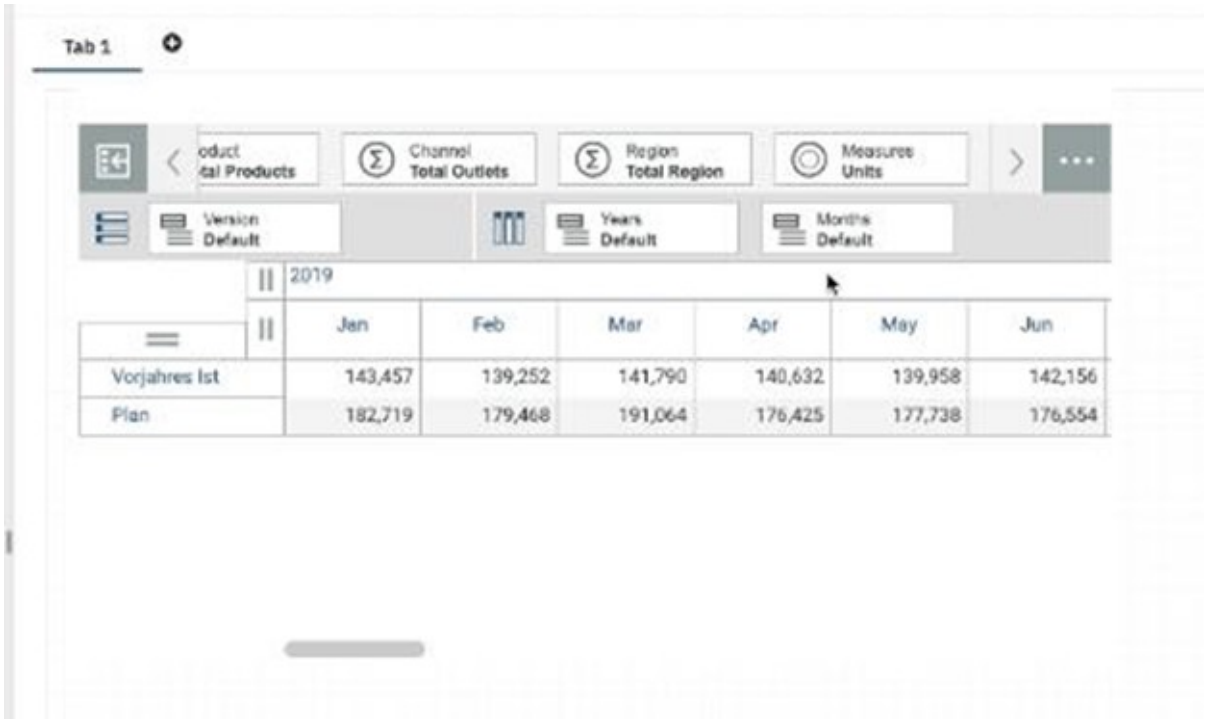

When you use multiple time dimensions in the column, then the product of the members must result in contiguous, evenly spaced periods of time with coarser grained elements that precede finer grained ones. In the following case, the inclusion of years, then months result in continuous time from 1995 on a monthly granularity.



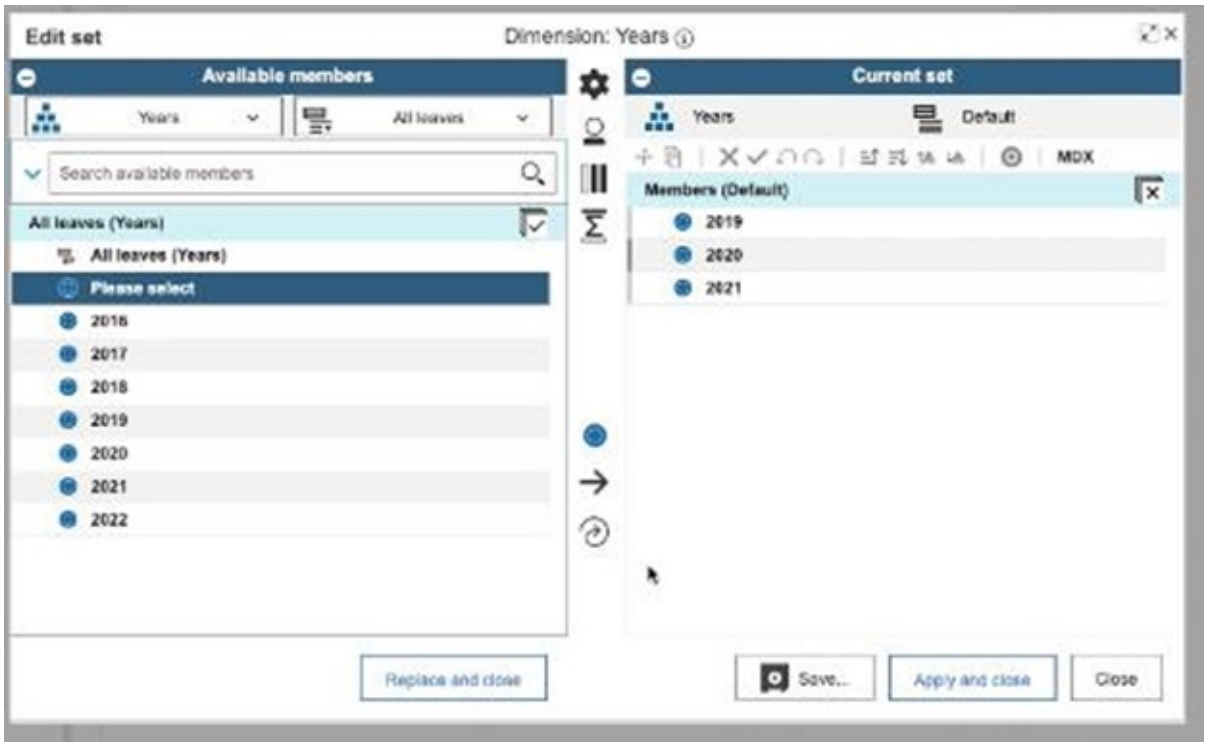

For time (that is, columns), other than the selected dimension, hierarchies, and the order that is placed, other aspects such as level of expansion or hidden and kept columns, do not have any meaning to time series forecasting. IBM Planning Analytics Workspace also sources historical data back as far along the time series as possible and source data from only the leaf members along the time series. One consequence of this is that leaves with multiple parents along the time series would result in non-contiguous time, so special care must be taken in these cases. If possible, it is advised to consider adding a special hierarchy for time series forecasting to support the required time configuration.

It is not enough to edit the view because the service might need to traverse the source cube to get more historical details.

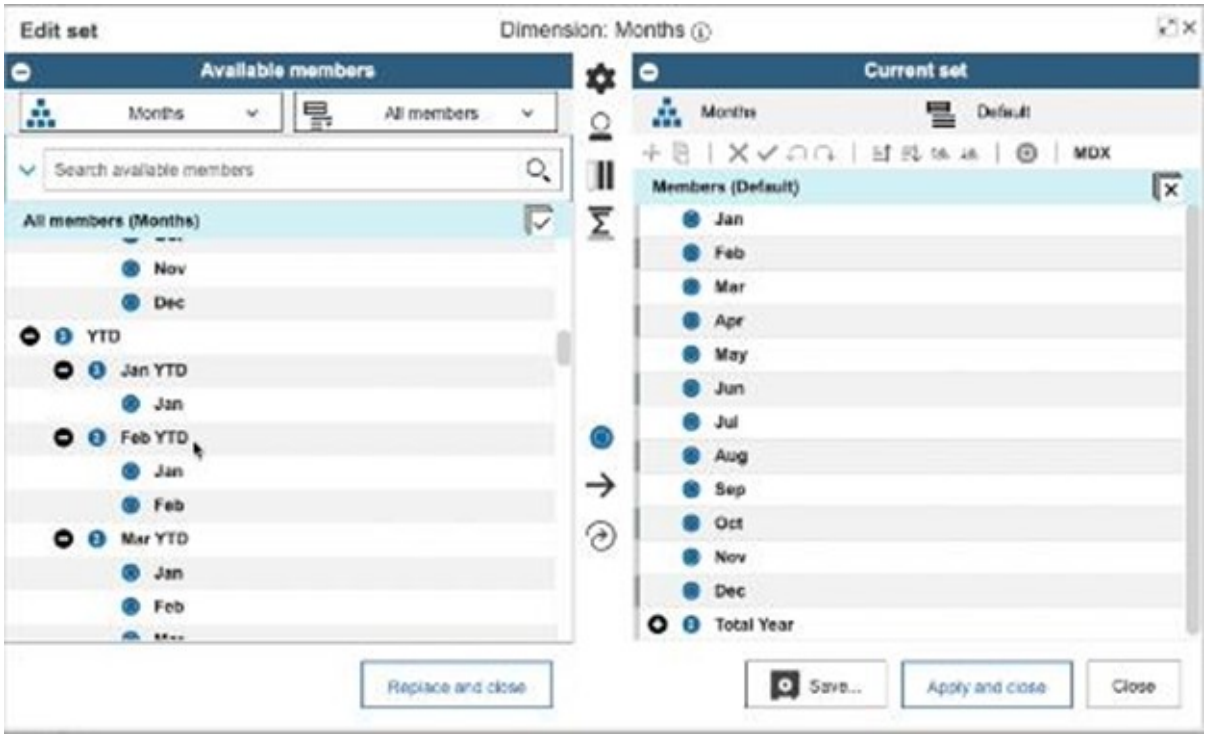
In this example, the time series is validly defined by nesting the *Years* and *Month* dimensions as columns.




The *Years* dimension is valid:



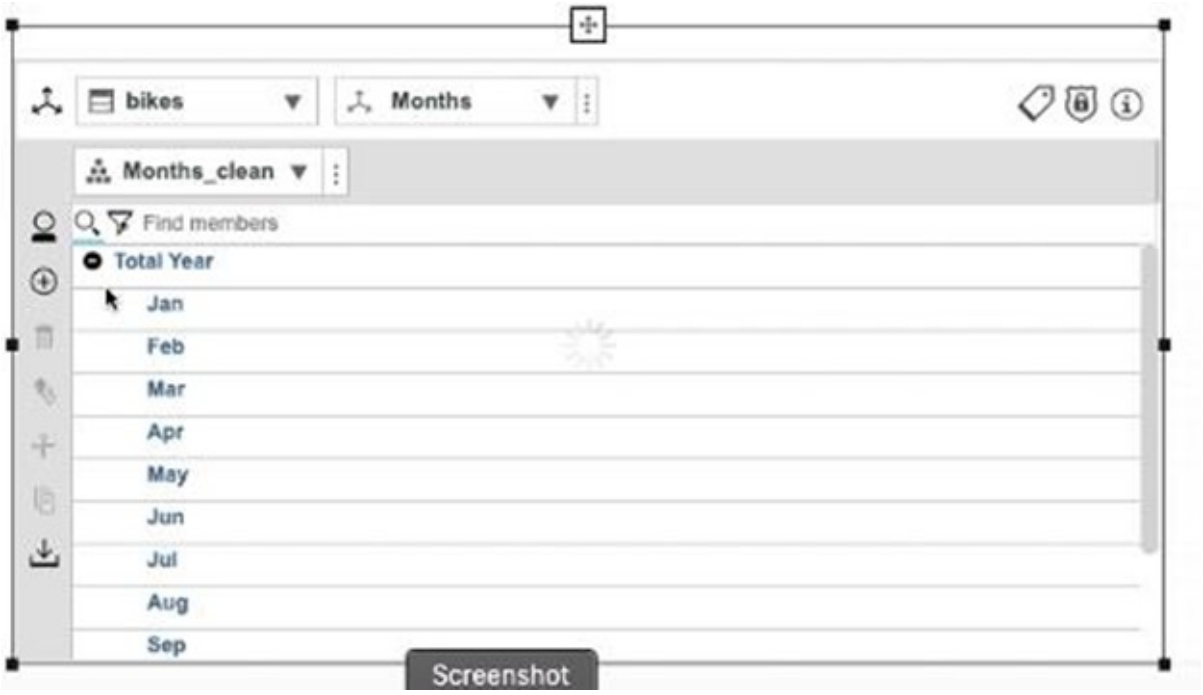

However, the month dimension was invalid:



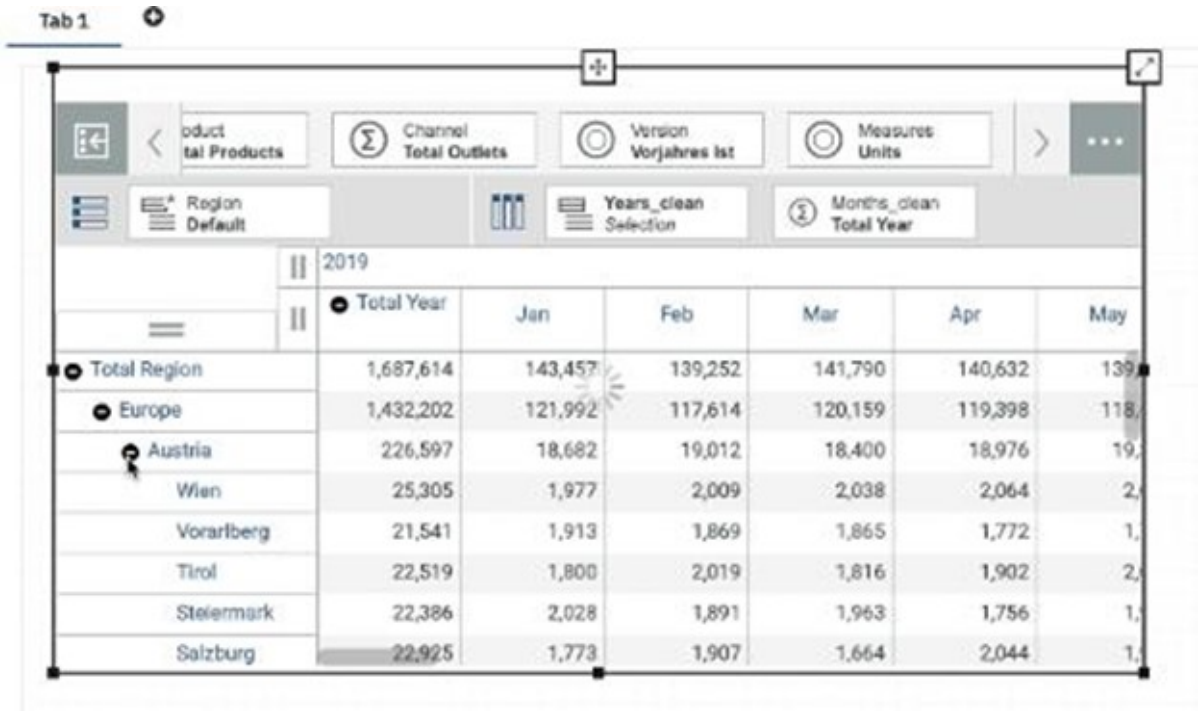

The leaf members of the *Month* dimension contain multiple summaries or parents. For example, *Jan*.

Second, the *month* dimension does not contain a single continuous time series, that the month members repeated.

A possible correction is to create a hierarchy that includes only the leaf months:




Correct time series are displayed next.




## Historical data

The quality and amount of historical data that is used for forecasting must meet certain requirements to be most effective. Planning Analytics Workspace reaches back as far as possible along the time dimensions that are configured on the column of the exploratory view from which the forecast is started from. Depending on the design of the dimension and nature of the data, the forecast might not be optimal. Some rules to keep in mind:

**Univariate forecasts**

The historical data must represent at least double what is being forecasted. For example, if the data has 3 years of data, then anything forecasted past 1.5 years must be disregarded.

The historical data must represent at least two seasons of data if it has seasonality. For example, if the data cycles yearly and the leaf time members are at a month level of granularity, then it must have at least 24 months of data in the history. This way, the feature can automatically detect the cycle.

**Multivariate forecasts**

At least 16 historical data points are required for the forecast to be successful.

Seasonality doesn't not apply for multivariate forecasts.

## Forecast data cells

The forecasting feature forecasts into all members that remain along the time series after your chosen start date. It does not create members in a time dimension. The members must exist. What this means is that they must also represent contiguous time. If any calculated cells exist along this series, for example, then the forecast fails since these cells cannot be written into.

## Version dimension

The version dimension, if you choose to use one, must appear in the context area of the exploration view. If the version dimension is placed in any other position on the view, it is not available for selection as target version on the **Advanced** tab when you set up your forecast.

If you want to see the values of all version members at once, create an independent view for this purpose.

### Zero suppression

Be aware of these considerations when using zero suppression in conjunction with forecasting.

**Zero suppression on columns**
Although zero-suppressed columns cannot be seen in the cube viewer, they might be required to store resulting forecast values. If you ensure that there are no nested columns that don't represent contiguous time segments (ie. years, quarters, months), you should not need to suppress zeroes on columns when forecasting.

**Zero suppression on rows**
If you have configured the **Advanced Tab** to specify where you want to save the predicted values, **do not** use zero suppression on rows. This can result in empty rows being suppressed in the target location where you want to save the predicted values. An error results, as there is nowhere to write the predicted values.

### Consolidated values

Although consolidated members are supported in the context area of the view from which forecasting is initiated, the resulting data spreading can consume significant resources. By default, forecasting uses proportional data spreading. If you want to spread data using a different algorithm, you can change this after forecasting.

### Variables for multivariate forecasts

Variables for multivariate forecasts must reside in a cube that is present on the same database as the view from which the forecast is initiated. Variables must exist in a cube that uses the same time dimension as the view from which the forecast is initiated.

### Test the setup with preview

An important last step in the setup is to preview the results of at least one time series to ensure that the results make sense. If they don't, you likely have something wrong with satisfying one or more of the requirements. The next section describes how to preview.

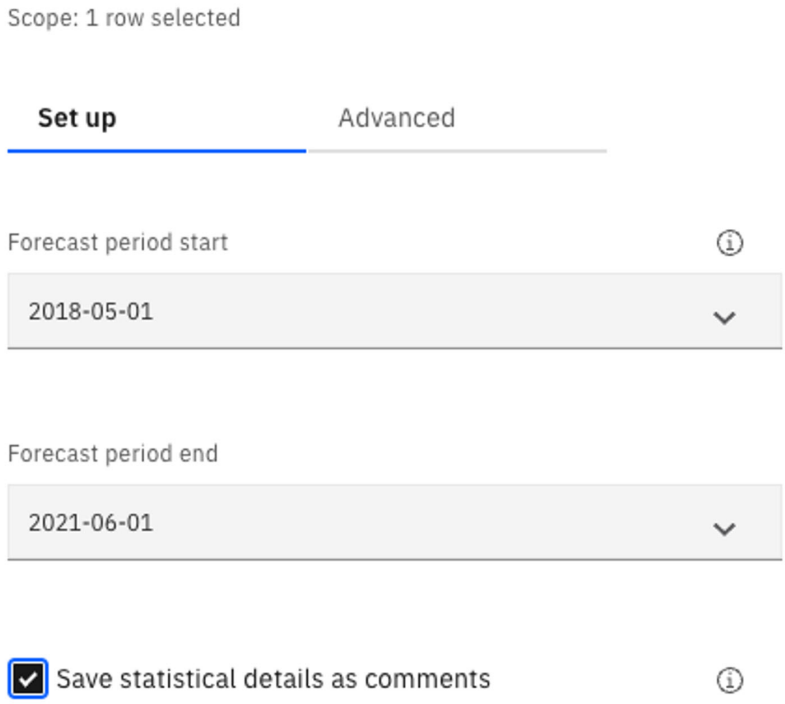
## Forecasting options

You can create or modify your forecasts by setting a time period, confidence interval, historical data, and data spreading in the **Forecast** dialog box.

Click the **Forecast** icon  and select **Univariate** or **Multivariate** to open the dialog box.

## The Set up forecast tab

Use the **Set up** tab to define the time period for your forecast. If you are generating a multivariate forecast, you'll also use this tab to define the variables for the forecast.




The following options, which are applicable to both univariate and multivariate forecasts, must be set on the **Set up** tab.

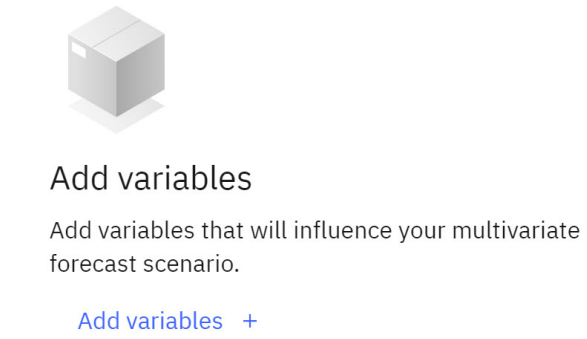
**Forecast period start**
The start of the forecast period. A period is the smallest time interval between neighboring points in the data.

**Forecast period end**
The end of the forecast period. The forecast period end field is populated automatically to reflect the last member on the dimension.

**Save statistical details as comments**
Enable this option if you want to save statistical details as cell comments. For performance reasons, the option to save statistical details as comments is only available if the scope of the forecast is less than 25 time series. For more than 25 time series, the option is disabled.

For a multivariate forecast, you must also specify the forecast variables. You can define up to three variables.

1. Click **Add variables**.




2. On the **Multivariate forecast** page, select the **Cube** that contains the variable that you want to use in your forecast.

   The list of available cubes is automatically filtered to include only cubes on the same Planning Analytics database that share the same time dimension as the cube for which the forecast is being performed.
3. On the **Selected members** panel, select the members that identify the variable you want to use in your forecast.

   The **Selected members** panel initially displays the default hierarchy for each dimension. If you want to select a member from a different hierarchy, click ⋮ on any dimension tile, then click **Replace hierarchy**.
4. Select the **Include future values** option if you want to include future values for the specified variable when generating a forecast.

   If the historical data include contains less than 64 data points, the future values for variables will not be considered in the forecast.
5. If you want to define a second variable, repeat steps 1 through 4.
6. If you want to define multiple variables, repeat steps 1 through 4.

## The Advanced tab

Use the **Advanced** tab to set the scope of historical data, define a confidence interval, define where you want the forecast values to appear, and set data spreading options for writing forecast data into consolidated members. Some options on the **Advanced** tab are not applicable to multivariate forecasts, as noted in the option descriptions.

Some of these Advanced options are not available to all user roles. When applicable, these restrictions are also noted in the option descriptions.

The following options are available on the **Advanced** tab.

**Seasonality**

Not applicable to multivariate forecasts. This option is disabled for consumer users.

If your model is highly seasonal, consider using a univariate forecast.

This option specifies the seasonality with which to build the model. Seasonality is when the time series has a predictable cyclic variation. For example, during a holiday period each year.

The default value is **Auto-detect**. **Auto-detect** automatically detects seasonality by building multiple models with different seasonal periods and choosing the best one.

If you switch **Auto-detect** off, you can specify seasonality by entering a non-negative integer, such as 0, 1, 2, 3 in the **Enter seasonality interval** field.

To specify a non-seasonal model, set the **Enter seasonality interval** to 0 or 1. A model with user specified seasonality is displayed only if the seasonal model is more accurate than all of the non-seasonal models.

**Select scope of historical data used**

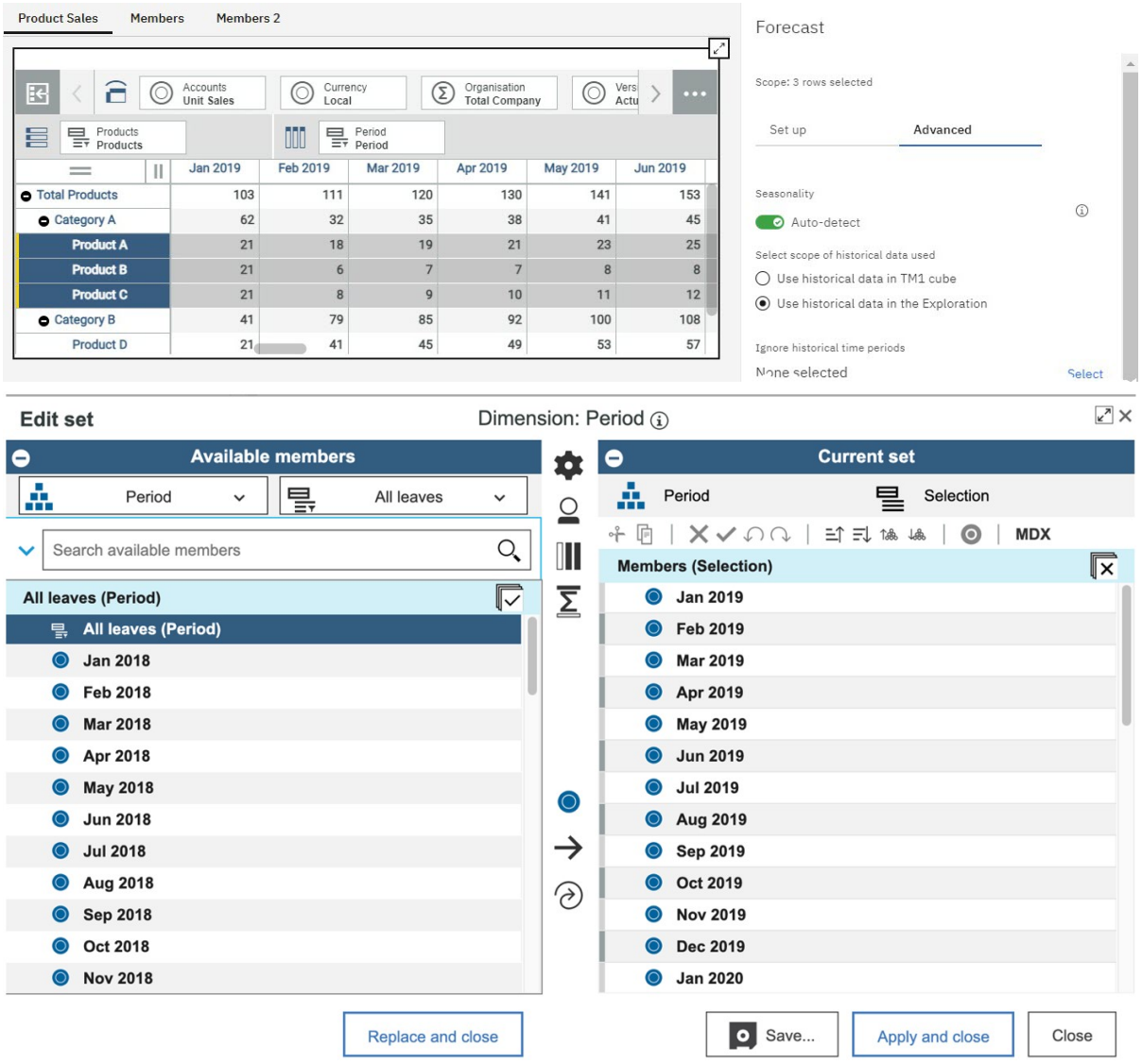
This option is disabled for users with the analyst or consumer user role.

Select between the following two options:

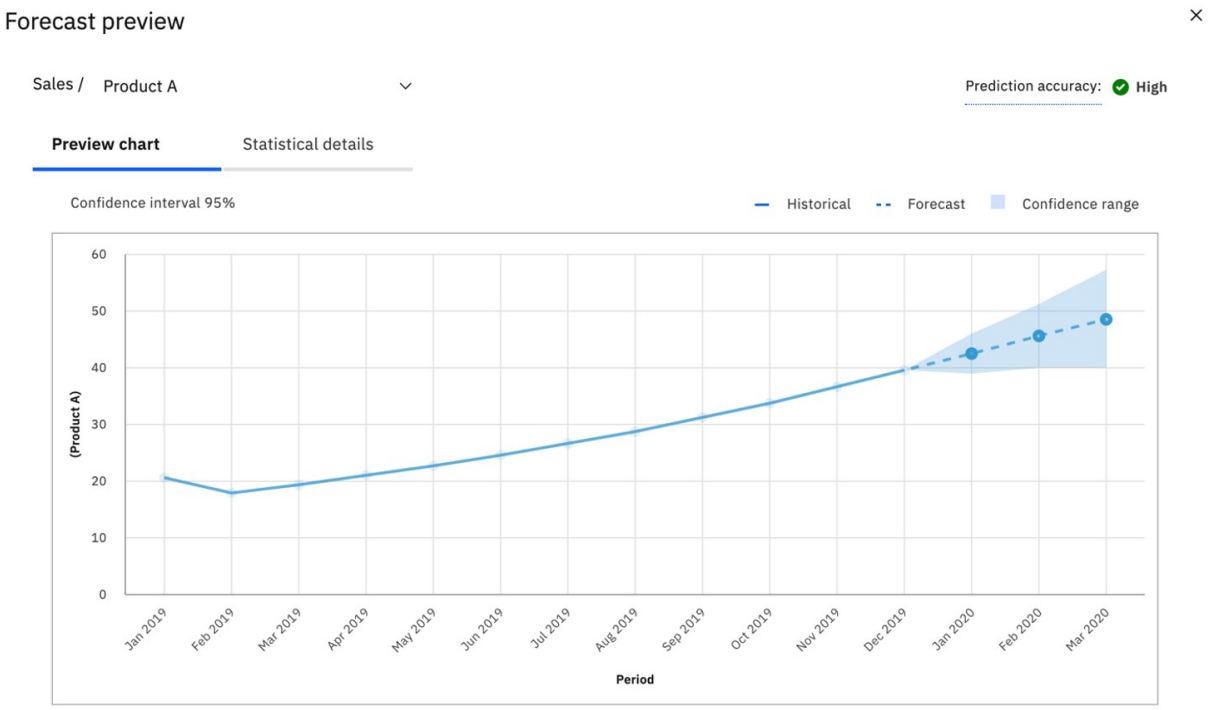
- **Use historical data in TM1 cube**
- **Use historical data in the Exploration**

The default option is **Use historical data in the Exploration** .

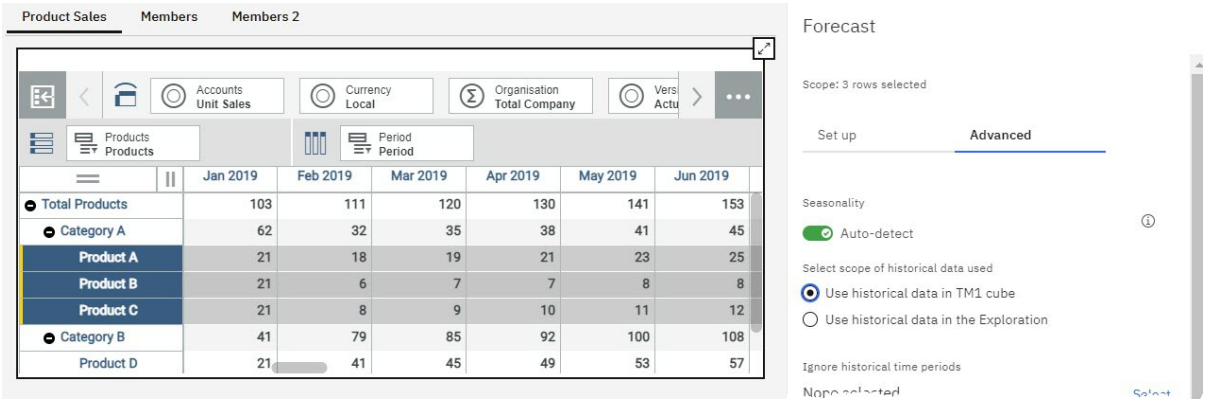
In this example, the leave members of *Period* hierarchy start at Jan 2018 and goes on to Sept 2021. However, the exploration view starts at Jan 2019.



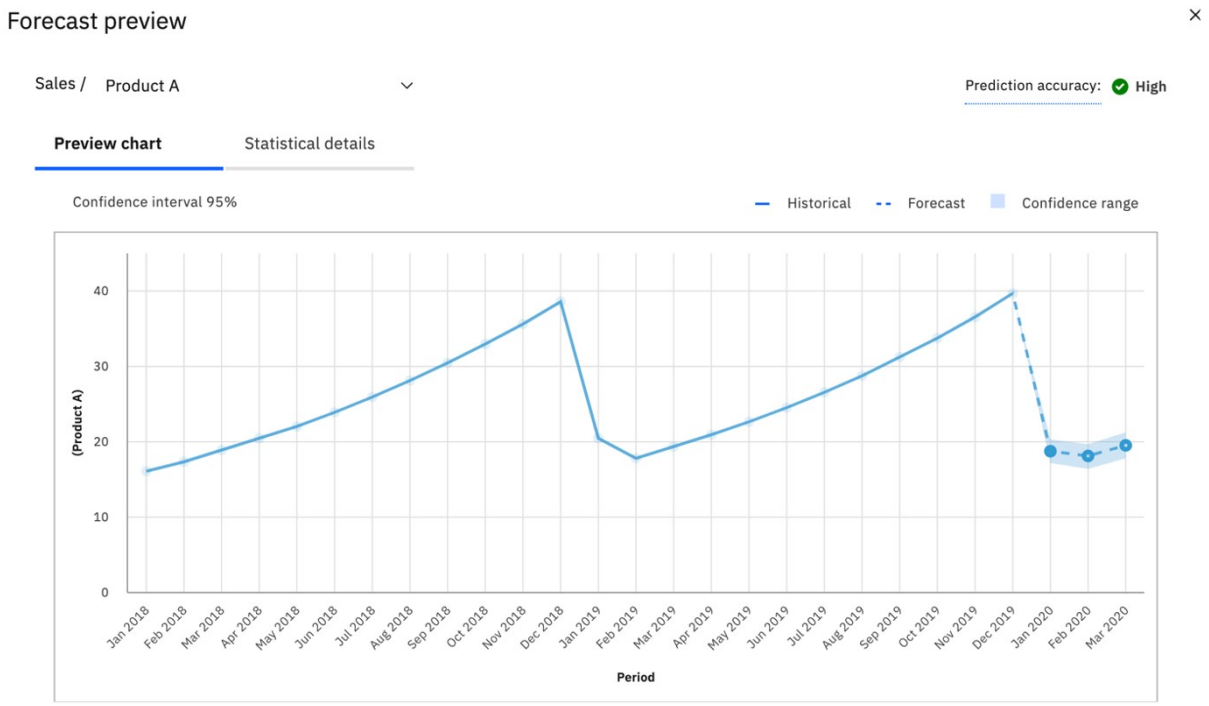

When the option **Use historical data in the Exploration** is selected, the historical data that is used for the prediction starts from Jan 2019 and includes only the members in the current Exploration view. As shown in the next image of the forecast preview, the visualization starts at Jan 2019, which is the start of the Exploration view and also used as the starting point for the history. Data for Jan 2018 - Dec 2018 is not included in the history of the prediction in this case.




Conversely, when you use the option **Use historical data in TM1 cube**, the leaf members of this *Period* hierarchy start at Jan 2018.




Based on this selection criteria, the forecast preview uses data from Jan 2018 onward to include all members in the Period dimension as the historical data. The following forecast preview image shows that the start point of the history is Jan 2018. Because more data is included as the history, and that data contains a decline, the forecast results changes.




**Select confidence interval**

This option is disabled for consumer users.

Not applicable to multivariate forecasts, which do not predict a range with high and low values.

The certainty with which the true value is expected to be within the specified range. You can see corresponding confidence interval in a tooltip by hovering over any forecast value. The confidence interval is displayed as higher and lower bounds.

You can select three different confidence intervals: 90%, 95%, and 99%. The default is 95% and the lower and higher bound define the range at which you can be 95% confident that the true value lies within that range.

**Adjust outliers**

When this option is turned on, all detected outliers are corrected before performing the forecast. You can preview the effect of this by selecting a row that contains an outlier and clicking the **Preview** button. See "Previewing with detected outliers" on page 548 for further information on how **Adjust outliers** works.

**Ignore historical time periods**

This option is disabled for consumer users.

Ignores a specified number of data points when building the model and computing the forecasts.

You can either ignore a set of data points at the end of a time series, or ignore individual data points in different places in a time series. The ignored data points are interpolated. A linear interpolation is used to populate the ignored values. Select the ignored periods in the nested **Ignore time periods** pane. You can select periods either as individual points or from a starting point onwards, until the start of the forecast.

**Note:** When you ignore historical time periods, the selected time periods in only time series are ignored. Time periods are not ignored in external variable time series.

By default, no historical data is ignored. If no missing values exist, then all of the historical data is used and the first forecast point is after the last historical data point.

Ignoring data period can be useful when the data is incomplete. For example, you might be doing a forecast halfway through a month. Exclude this month from the forecast by setting **Ignored historical time periods** either to that specific month, or to the last data point if that month is your last.

**Spread forecast values**

Define the data spreading method that you want to use to spread data from consolidated cells.

You can use the Proportional and Relative proportional data spreading methods. For more information, see “Apply data spreading to a forecast” on page 538.

**Accepted range**

There might be circumstances where you want to limit the maximum and minimum values for a forecast. For example, your business likely cannot produce a number of units in excess of manufacturing capacity or sell a negative number of units, though a forecast might predict such values. To address these circumstances, the **Accepted range** section let's you set the maximum value accepted and minimum value accepted for a forecast.

The **Maximum value accepted** specifies the highest value that can be saved for the forecast. For example, if you specify 100 as the maximum, any forecast values that exceed 100 will be saved as 100.

The **Minumum value accepted** specifies the lowest value that can be saved for the forecast. For example, if you specify 0 as the minimum, any forecast values that are lower than 0 will be saved as 0.

**Important:** These settings apply only to the data that is saved for a forecast. The forecast preview will still show the actual generated forecast values.

**Where do you want to save the predicted values**

This option is disabled for users with the analyst or consumer user role.

This section lets you identify the dimension, hierarchy, and member where you would like to save your forecast values. For example, you would typically forecast from the ‘Actual’ member of the ‘Version’ dimension into the ‘Forecast’ member of the ‘Version’ dimension.

For univariate forecasts, you can also specify the members where you want to save upper-bound and lower-bound forecast values. If you want to save the upper-bound and lower-bound values for consolidations, select the **Save upper and lower bound for consolidations** option. This option is not available for multivariate forecasts, which do not generate upper-bound and lower-bound forecast values. This option is not available to consumer users.

## *Apply data spreading to a forecast*

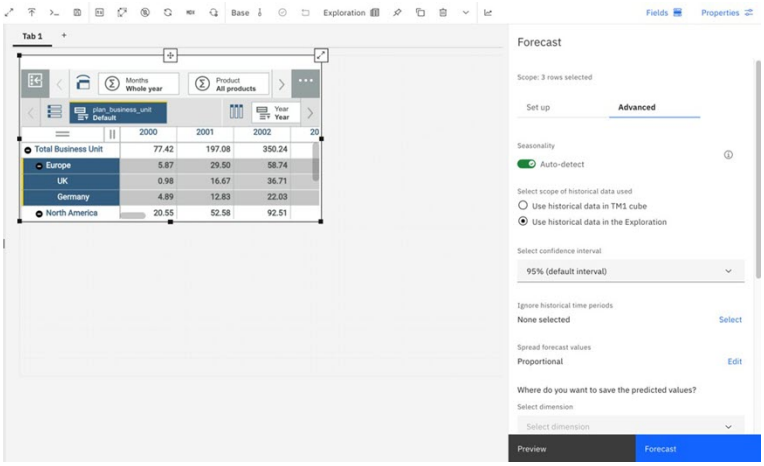
For predictions that are written to consolidated cells, you can use either the Proportional (default) or Relative proportional data spreading methods.

**Note:** Data spreading in a forecast requires TM1 server 11.8.7 (PA 2.0.9.8) or later. If Planning Analytics Workspace is connected to a TM1 server prior to 11.8.7, the data spreading option for a forecast is not available.

See “Proportional data spreading method” on page 1256 for more information on Proportional data spreading.

See “Relative proportional data spreading method” on page 1262 for more information on Relative proportional data spreading.

The following example shows how you can define and use data spreading. In this case, we are performing a forecast on three time series on the context: UK, Germany, and the consolidated member Europe. If all items on the context were *not* consolidated, then data spreading would be applied only to Europe. However, in the following scenario, given there are consolidated items on the context ('All Products' and 'Whole year'), then all three time series use the selected spreading option.




Click the **Edit** button for the **Spread forecast values** option, then select the type of data spreading you want to use on the **Spread data options** page.

**Proportional** - Proportional data spreading distributes a specified value among cells proportional to existing cell values. When you select the Proportional method, no further configuration is required. You can click **Apply** to set the data spreading method.

**Relative proportional** - Relative proportional spreading distributes values to the leaves of a consolidation cell proportional to the leaves of a reference cell or cells. When you select the Relative proportional method, you must define the **Reference type** for the spreading operation. Select one of the following reference types.

**Seasonality (Auto)**
: For each time series, after the prediction is made but before the forecasted values are written, the forecasting process attempts to identify a seasonality period. If one is identified (for example, for a retailer with a month granularity time series, may be the number 12, representing a yearly, repeated cycle of ups and downs in the data), then the reference cell used for the relative proportional spread is 12 cells back along the time series. If no seasonality is detected, then the last historical cell is used for reference.

**Last historical time period**
: For every consolidated cell written to (that is, forecasted values along a time series), the reference cell for the relative proportional spread is the latest non-ignored value before the Start Date along the time series. On a cell-by-cell basis, the behavior is identical to that of doing a relative proportional spread on that cell manually with the aforementioned reference cell.

**Specific time period**
: For every consolidated cell written to (that is, forecasted values along a time series), the reference cell for the relative proportional spread is the one you select with the **Reference time period** option for each given time series. On a cell by cell basis, the behavior is identical to that of doing a relative proportional spread on that cell manually with the aforementioned reference cell

**Range of time periods**
: This works similar to the **Seasonality (auto)** reference type in that the reference cell cycles on a range along the time series. Namely, for every consolidated cell written to (that is, forecasted values along a time series), the reference cell for the relative proportional spread is the one identified within the range defined by the **Start of time period** and **End of time period** that you specify. When the specified **End of time period** is reached, the next reference cell will be the 'Start of time period' and the cycle repeats until all forecasted cells are written to.

  Click **Apply** after you define a reference type.

## Forecasting preview

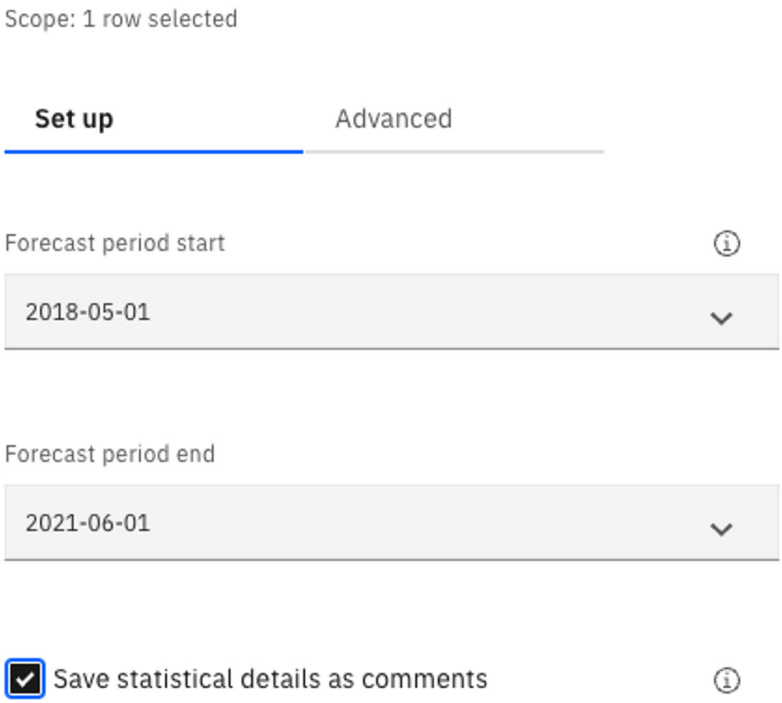
When the forecast options are set up, you can preview the forecast.

## Before you begin

**Note:** The preview operation does not change your actual data.

You can preview only one time series at a time. The forecasting preview shows what the results for the forecast look like, based on your input. This gives you the opportunity to analyze and work with projected forecast data before submitting it to the cube (database), helping you make more informed decisions.

By default **Preview** is not active. Select up to 25 rows and you are able to click **Preview**.

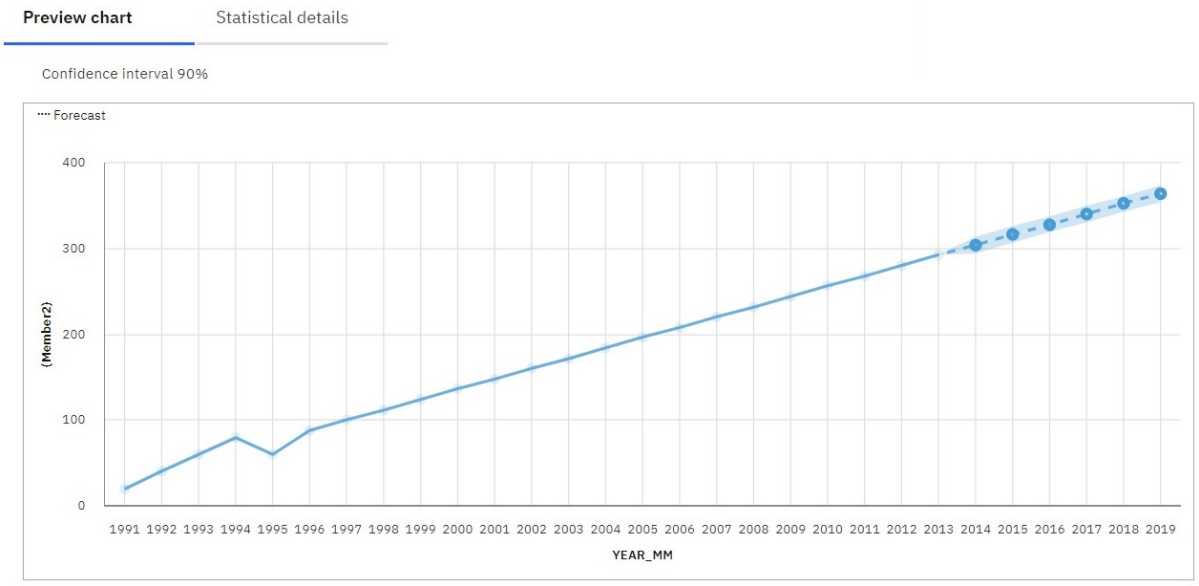



## Procedure

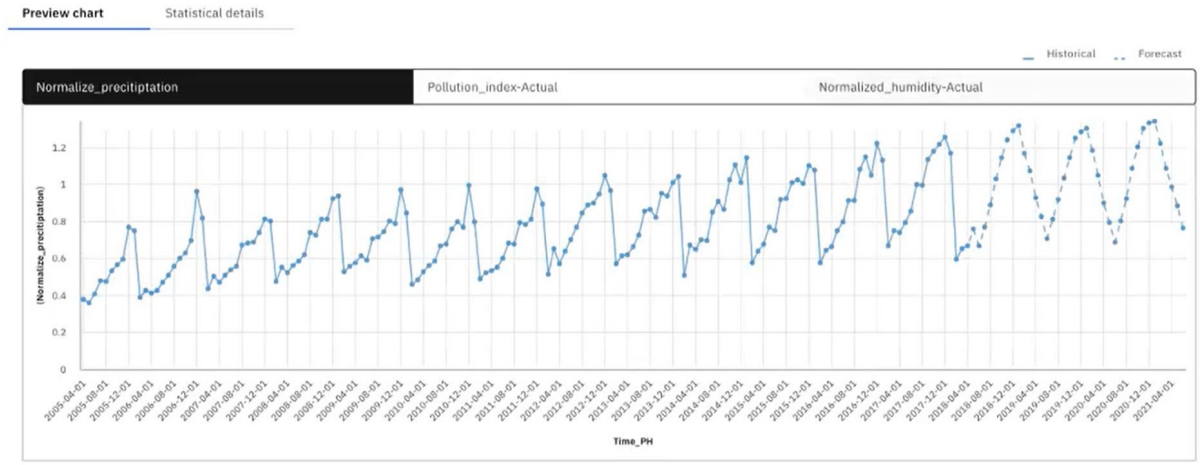
1. On the **Forecast** dialog box, click **Preview**.

   The preview is displayed with a visualization of the forecasted data, which is represented as a dotted line. Member captions, if available, are displayed on the preview by default.

   A univariate forecast shows predicted values, as well as a confidence envelope that indicates high and low bounds within a shaded area.




A multivariate forecast shows predicted values without a confidence envelope. A multivariate forecast also includes separate tabs for each variable included in the forecast. Click a tab to review data for the variables. A multivariate forecast does not include a confidence envelope with upper- and lower-bound values.




The **Prediction accuracy** shows: low, medium, or high.

2. Click the **Statistical details** tab to review the details for the forecast.

## What to do next

To export forecast values from the preview chart, click **Download data as CSV**.

The resulting CSV file contains data exported into the following columns:

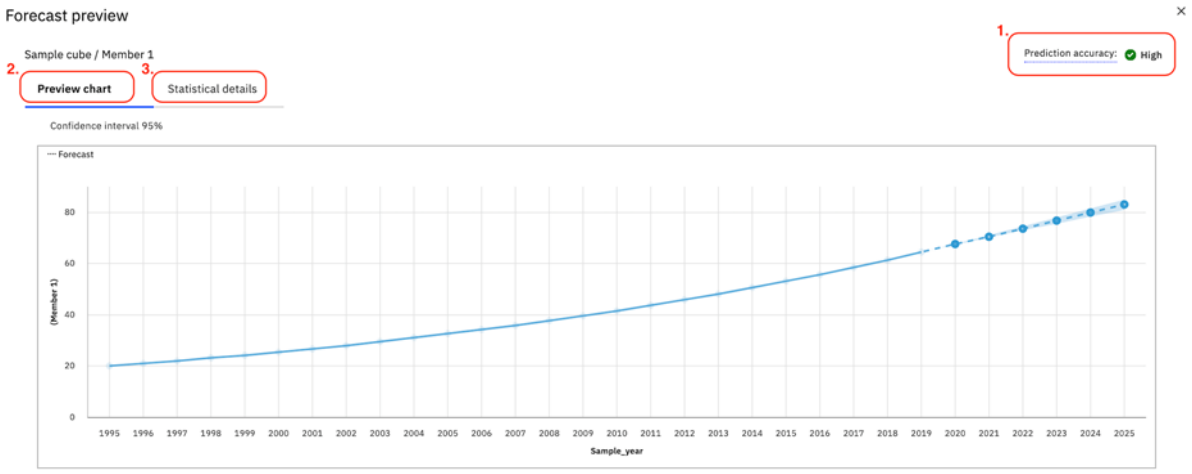
1. Name of the tuple or the selected row.
2. Forecast periods from the time dimension.
3. Forecast slice, which consists of forecast values and, for univariate forecasts, a confidence envelope that indicates high and low bounds within a shaded area. Multivariate forecasts do not include the confidence envelope.
4. Actual values for each of the slices and the corresponding forecast time period.

## Overview of the forecasting preview

The preview of the forecast consists of three main components.

The three main components are:

1. Prediction accuracy indication.
2. Preview chart. In a multivariate forecast, the preview chart includes tabs for each variable that is included in the forecast.
3. Statistical details, which vary for univariate or multivariate forecasts.




The illustrations on this page show a univariate forecast, but the concepts are the same for a multivariate forecast.

## Prediction accuracy




**Prediction accuracy** is one of the most important indicators of the usefulness of a given prediction and represents how well the model was able to fit the historical data. If the accuracy is indicated as low, then the exponential smoothing model that is identified was not a good fit for this data. The prediction is likely not usable and other means should be employed to derive this data.

## Preview chart

The preview chart consists of the following three main areas:

1. The confidence interval (1). This is not included in multivariate forecasts.
2. All historical information that was used to create the model.
3. And the forecasted data points based on the model that best aligned with the historical data.

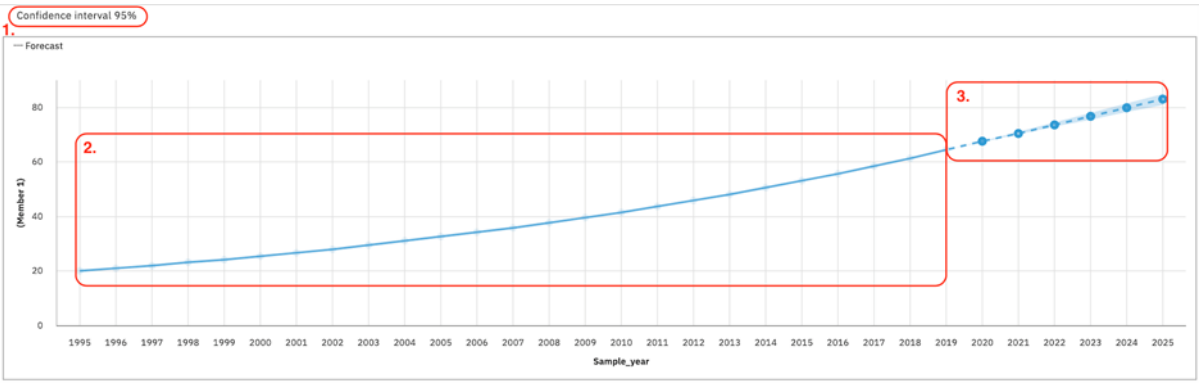



*Figure 1. Univariate forecast preview*

A multivariate preview also includes tabs for each of the variables that are used to calculate the forecast. Click a tab to view details.

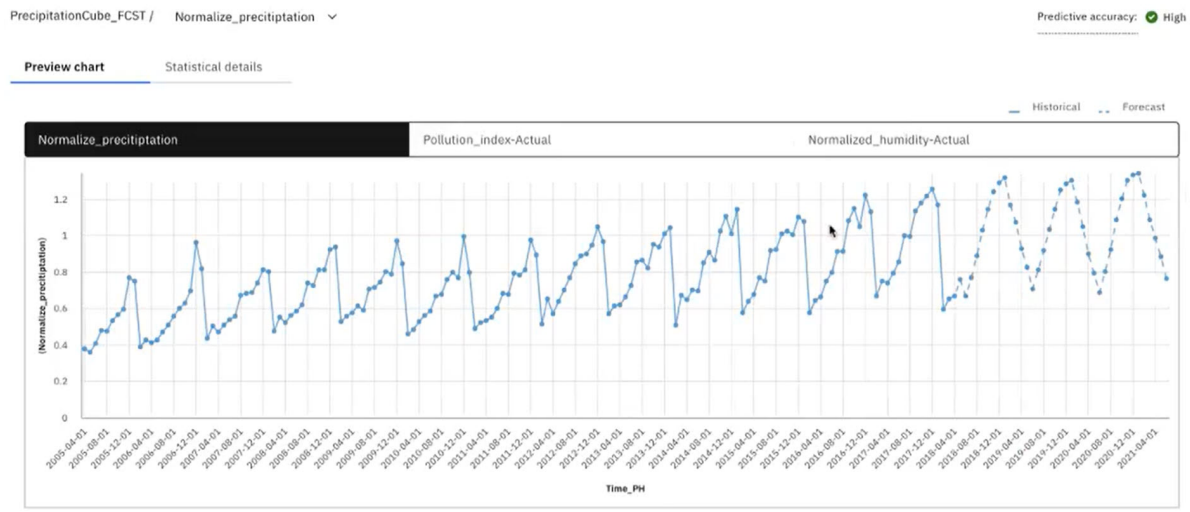


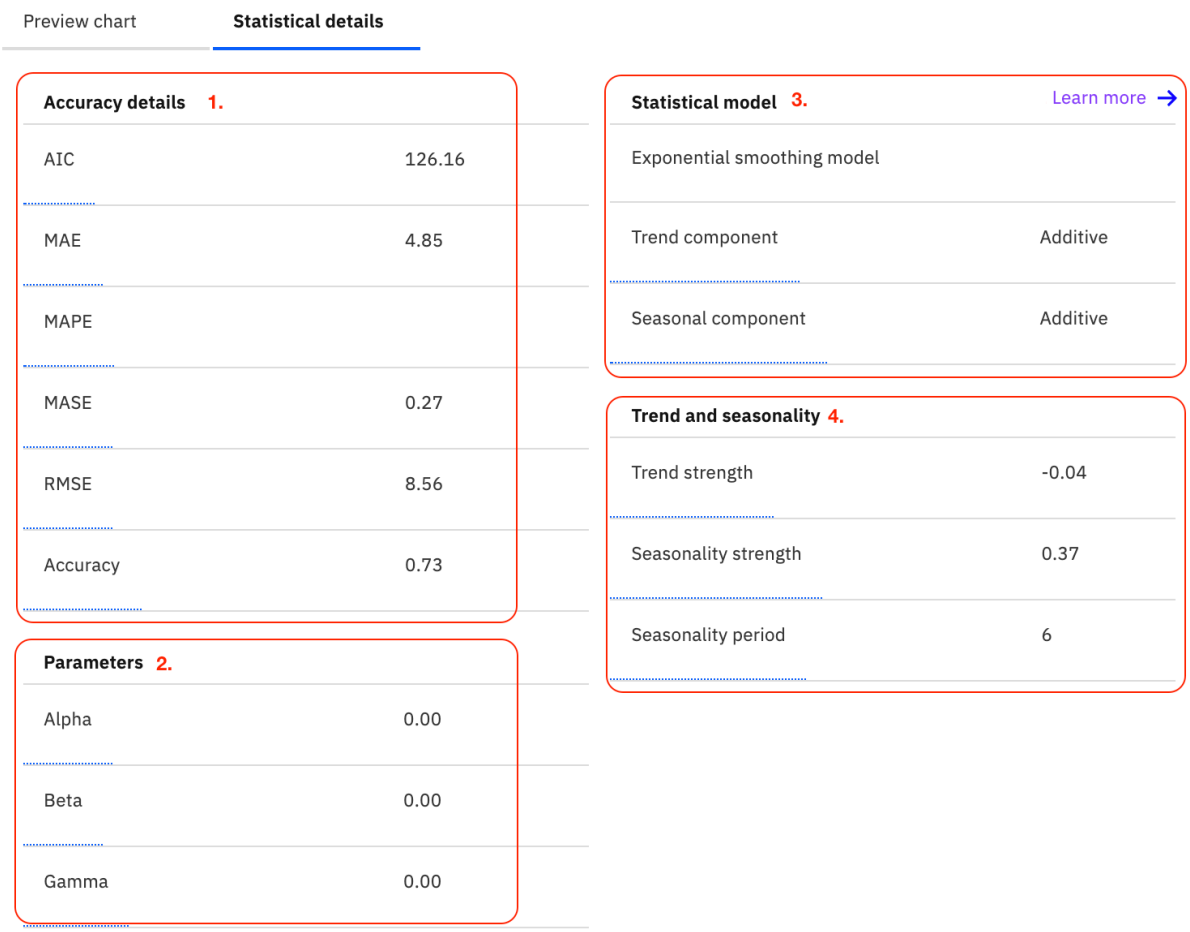

*Figure 2. Multivariate forecast preview*

## Statistical Details - univariate forecasts

The statistical details for a univariate forecast includes information that is typically useful to a data scientist and consists of the following five main areas:

1. The accuracy details, which include the number that is used to derive the overall accuracy of the forecast.
2. Parameters.
3. Information about the forecasting method that was used along with the detected trend and seasonality components.
4. Trend and seasonality strength and seasonality period. Period indicates information that relates to the cycles detected in the data along the time series.

If either the trend component or the seasonal component is **None**, then the respective characteristic was not detected and does not provide a **Strength** value (or **Period** for seasonality).




See “Univariate statistical details” on page 561 for more information.

## Statistical details - multivariate forecasts

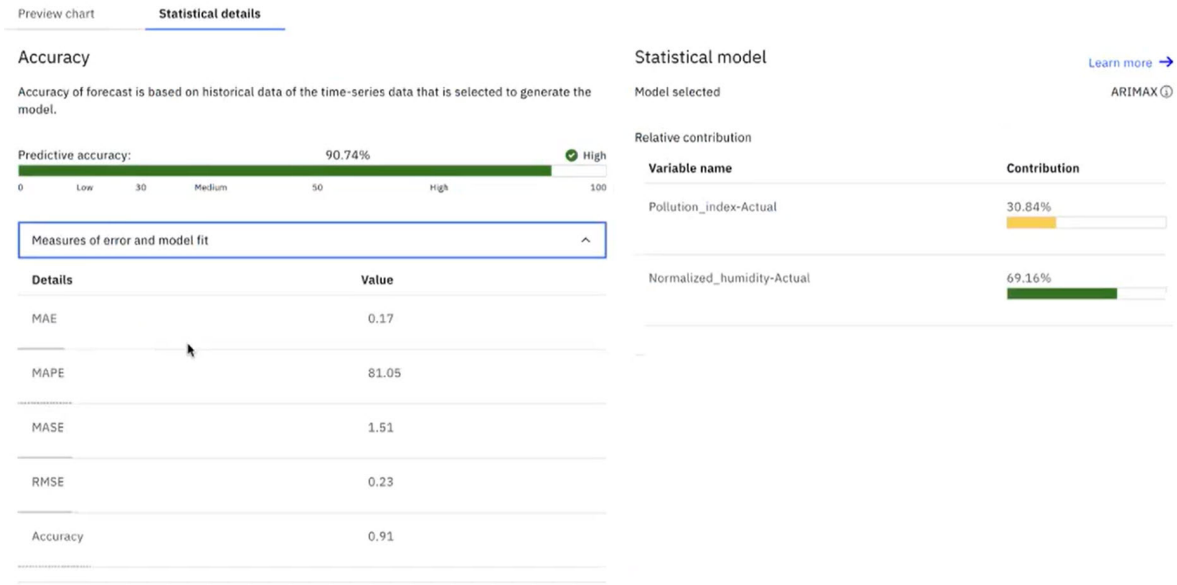
The statistical details for a multivariate forecast consists of the four main areas.

**Accuracy**
Indicates the accuracy of the forecast based on available historical data.

**Measures of error and model fit**
Provides details of the factors that contribute to the accuracy prediction. For more information, see “Multivariate statistical details” on page 564.

**Statistical model**
Indicates which model was used to generate the forecast. See Multivariate forecasts for more information about how a model is chosen for a forecast.

**Relative contribution**
Lists the variables that are used to generate the forecast and indicates the relative contribution of each variable to th foirecast.




## Visualization types that support forecasting

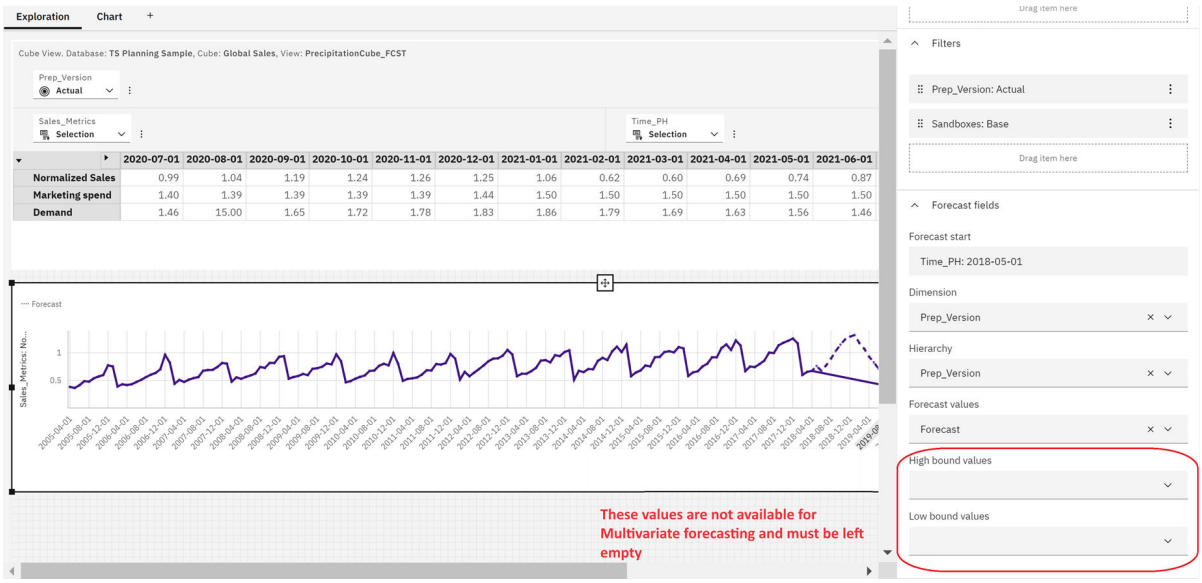
Forecasting is supported in line visualizations.

This table describes how forecasting features are represented in a visualization.

| Forecasting feature | Line visualization |
|---|---|
| Forecast points | Open circle |
| Confidence interval display* | Shaded region |
| Activate confidence interval* | Click any point |
| Number of confidence intervals displayed* | All |

* not available in multivariate forecasts

As indicated in the table, there is no confidence envelope for multivariate forecasts. However, the **Fields** panel for the visualization line is generic for both univariate and multivariate forecasts and contains **Low bound values** and **High bound values** fields. Both of these fields should be left empty for multivariate forecasts.



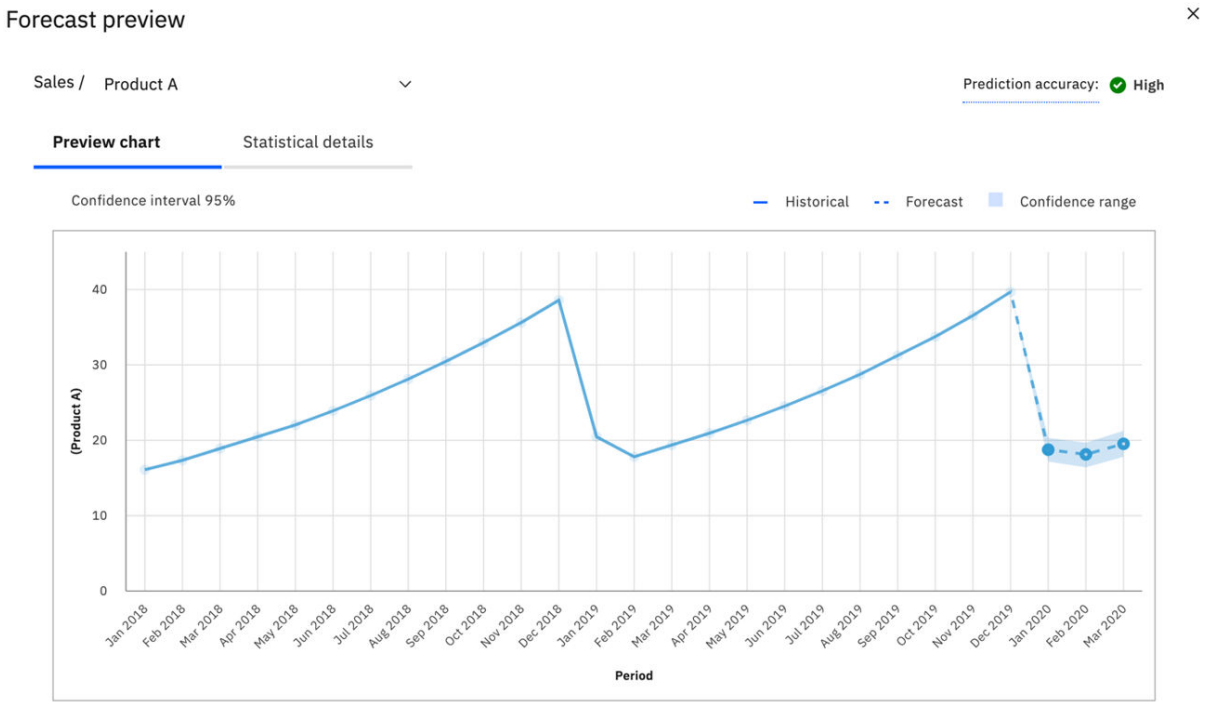

## Previewing more than one row

You can select up to 25 rows when you preview a forecast

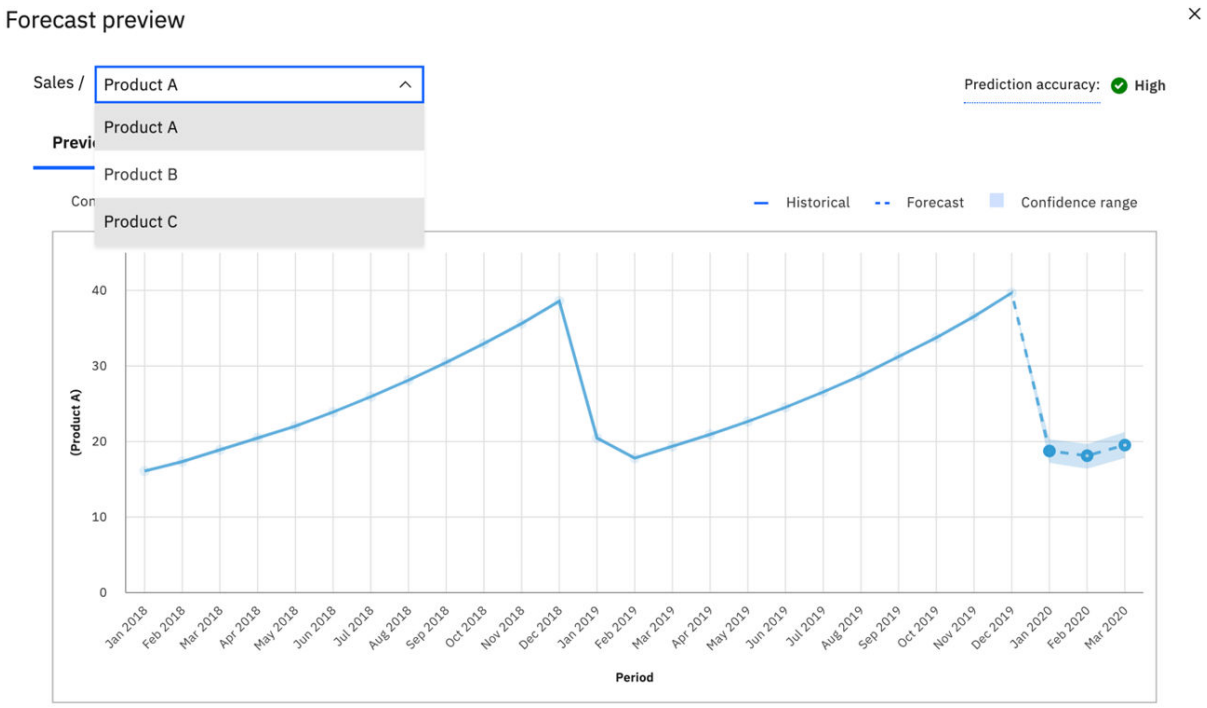
The illustrations on this page show a univariate forecast, but the procedure is the same for a multivariate forecast.

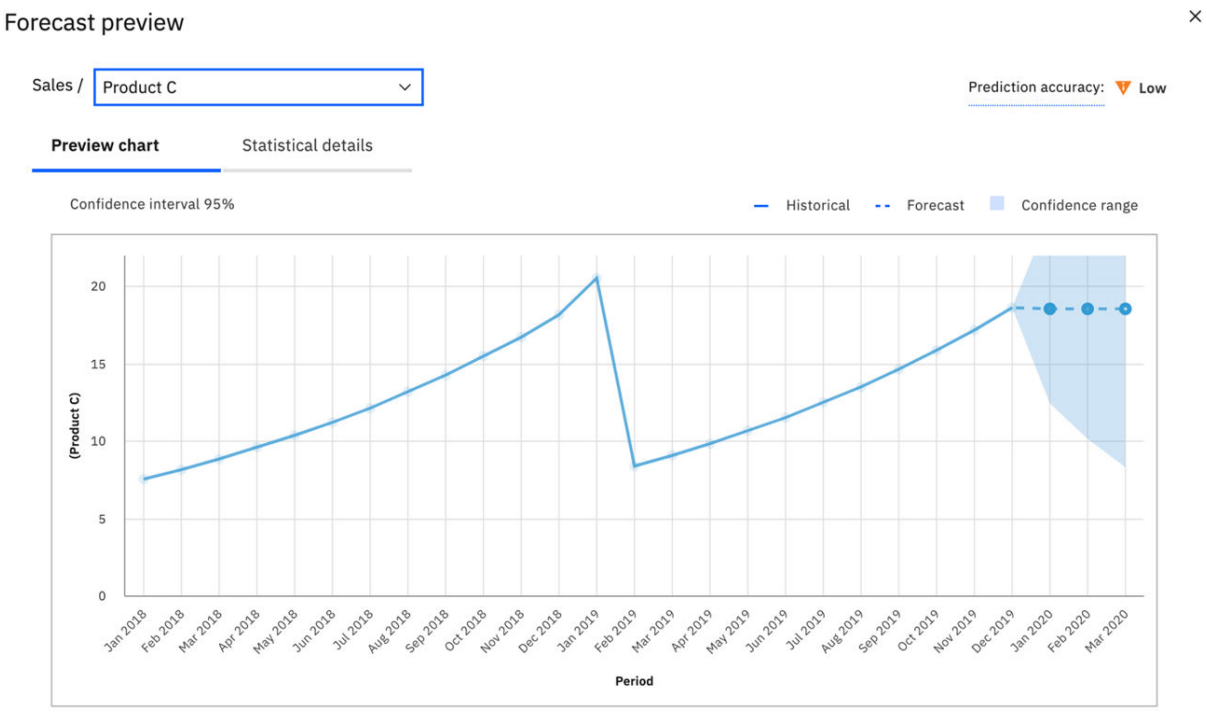
If more than 25 rows are selected, then **Preview** is disabled.

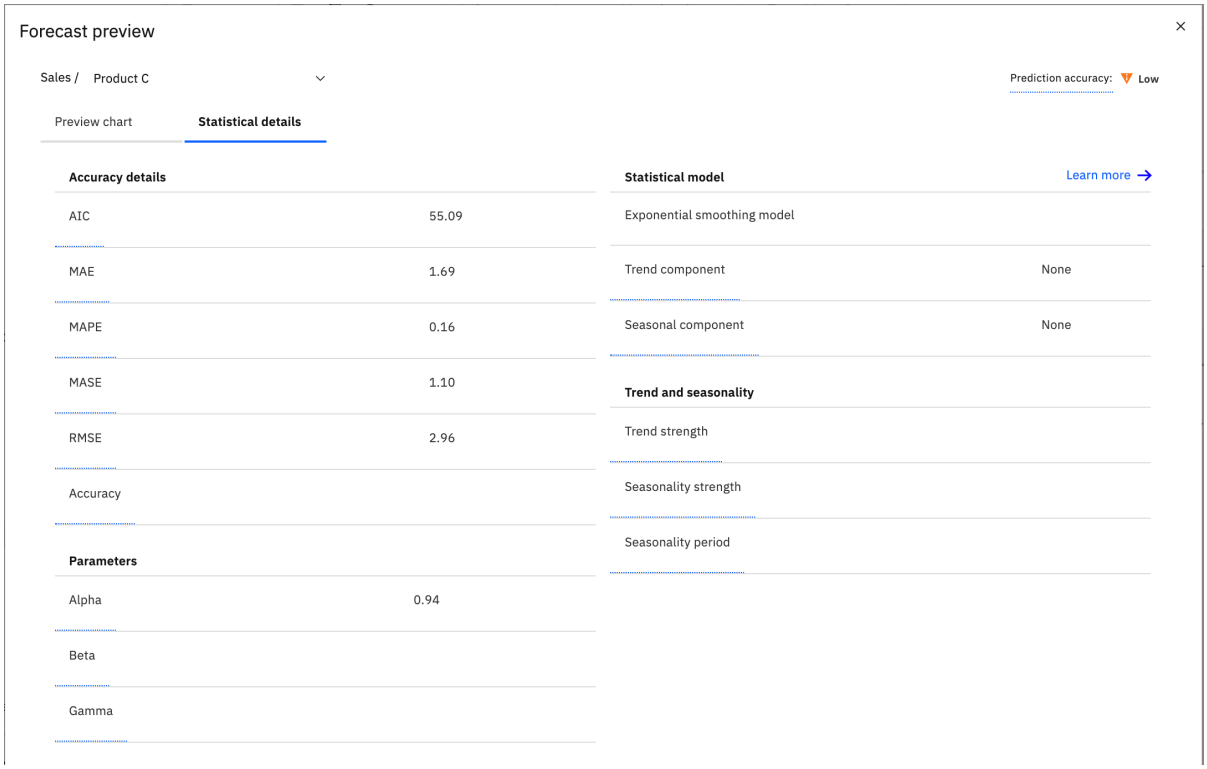
When you click **Preview**, the **Forecast preview** window opens. The results of the first row are displayed in the visualization and its statistical details. From the drop-down menu, you can select any of the rows included in the forecast.




When you select another item in the drop-down menu, the forecast runs for that item. A loading icon indicates that the forecast is running. When the forecasting is complete, the visualization, statistical details, and prediction accuracy are updated with the results of the selected item.

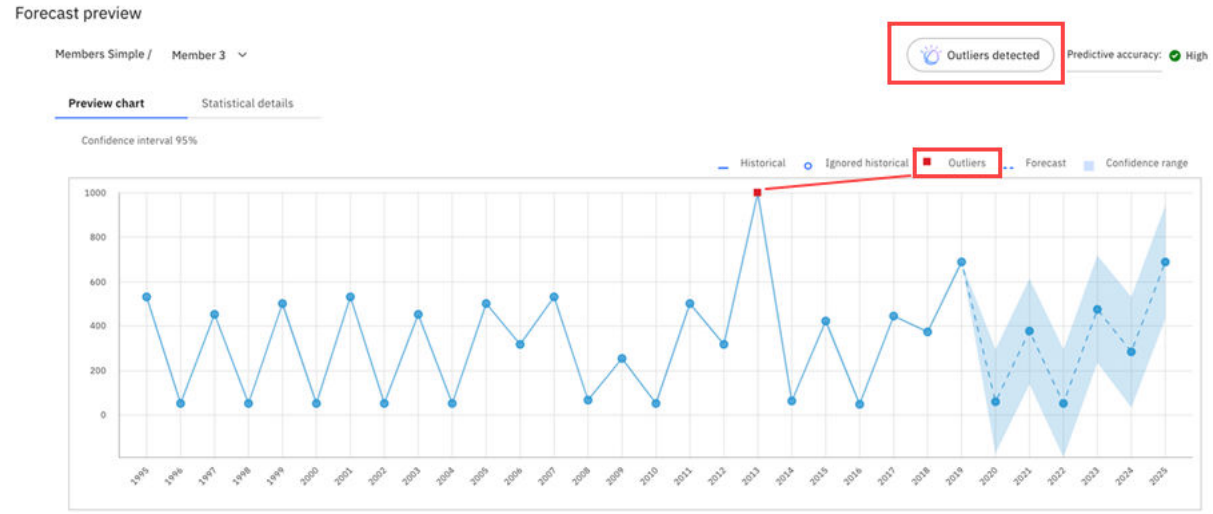

Forecast preview
Sales /
Product A
Product A
Product B
Product C
Prediction accuracy:
High
Historical
Forecast
Confidence range
(Product A)
Period


Forecast preview
Sales /
Product C
Prediction accuracy:
Low
Preview chart
Statistical details
Confidence interval 95%
Historical
Forecast
Confidence range
(Product C)
Period




## Previewing with detected outliers

When the **Adjust outliers** option on the **Advanced** tab is turned off, you are alerted of outliers that are detected in a preview by an **Outliers detected** button and highlighting within the graph that shows the outliers.

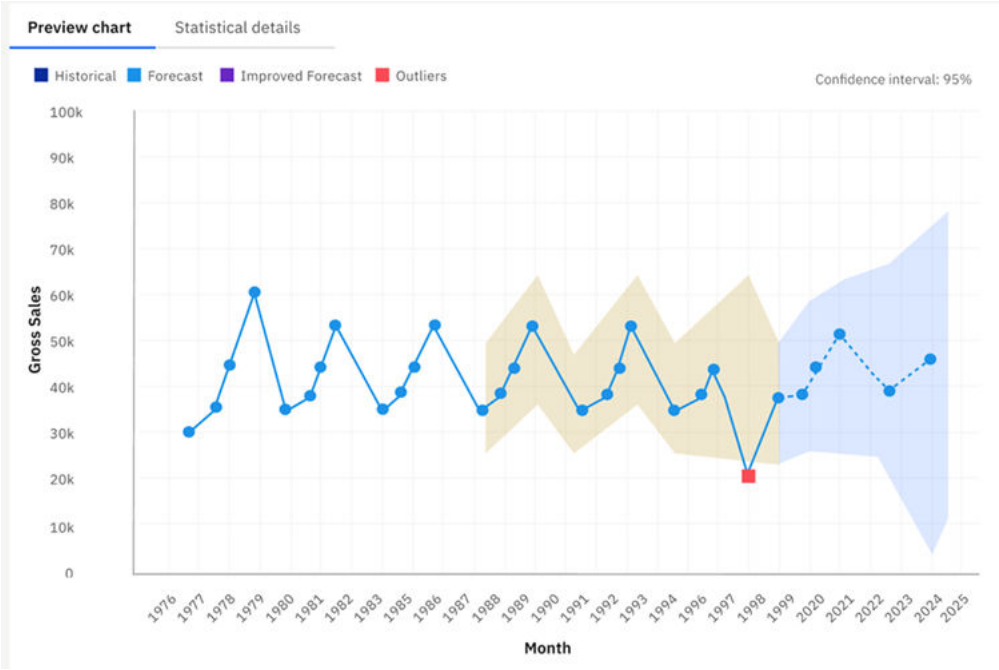



### *Outliers explained*

An outlier is a point in the last 20% of the historical data that is outside of the confidence range at that point in history.

When forecasting, many models are evaluated to determine which one proves to be most accurate overall, then the one with highest overall accuracy is selected. Although not displayed in Planning Analytics Workspace, the points along the model history have a confidence interval similar to that displayed with the forecasted values. Where many points along the model history may map near exactly to the actual historical points received, there may be a subset that sit outside of the confidence level

and it is these points that are highlighted as outliers. This is illustrated here, where you can see a single outlier.




With the confidence bound of the historical data super-imposed onto this chart, we see that, where the value was expected to be ~55k, we have a value of ~20k. The value is outside of the confidence bound, resulting in it being flagged as an outlier.

**Note:** Note: As outliers are corrected, the model changes and new outliers can result. This is explained in detail in Interpreting the preview.

### *Correcting outliers*

When Planning Analytics Workspace identifies outliers in your data, you can adjust a forecast preview to correct the outliers.

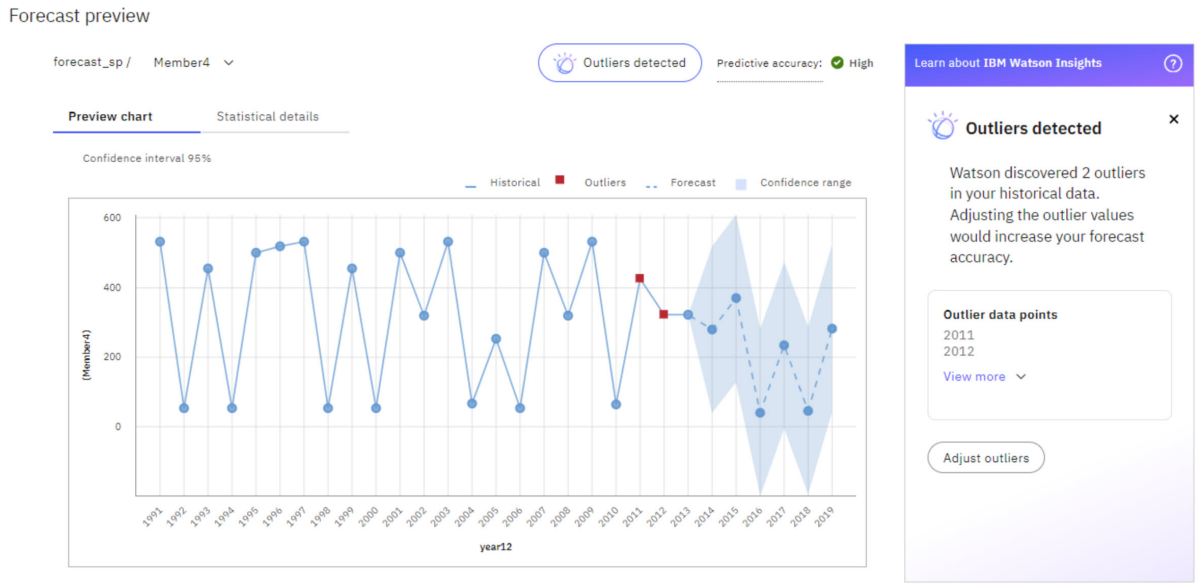
## About this task

When you choose to adjust for outliers in your forecast, the forecast model is corrected by using a "normalized" data point for the outlier to increase the forecast accuracy. The actual data point that is identified as an outlier is not modified on your Planning Analytics database.

## Procedure

1. Click the **Outliers detected** button on the preview.

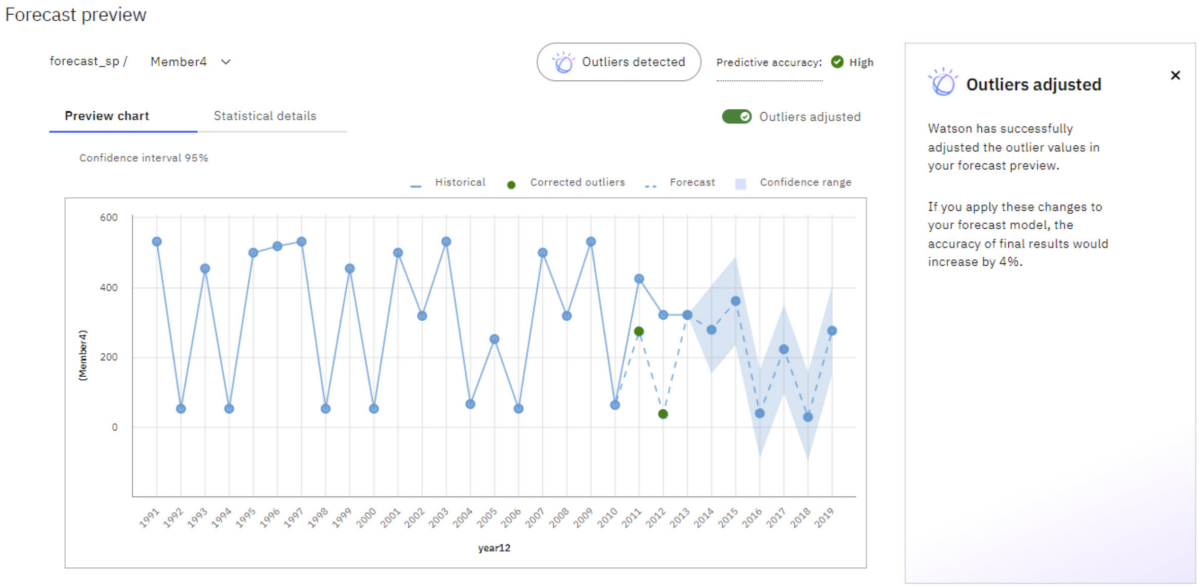
   The **Outliers detected** pane opens.




In this example, outliers are detected at the 2011 and 2012 data points.

2. Click the **Adjust outliers** button on the **Outliers detected** pane.

   The outliers are adjusted, which has an impact on all following values. The **Outliers adjusted** pane explains the impact of the adjustment. In this example, the adjustment results in a 4% increase in the accuracy of the forecast.




When multiple outliers are detected, they are adjusted sequentially from the beginning of the data set. This means each adjusted outlier value is applied to the calculation of the adjustment for the subsequent outlier, resulting in a more accurate forecast.

3. After you adjust outliers, an **Outliers adjusted/Outliers not adjusted** switch is available on the forecast preview. You can toggle the switch to see the forecast with or without the outlier adjustment. The **Outliers adjusted/Outliers not adjusted** switch is active on both the **Preview chart** and **Statistical details** tabs, so you can see the impact of outlier adjustment graphically and statistically.

## Interpreting the preview

When you preview forecast data for the first time, sometimes the results can be unexpected. Here are some situations where you might encounter unexpected preview results and explanations for why they can occur.

### Failure to forecast

#### *Not enough data points provided*

To perform a prediction, typically a minimum of seven historical data points must be provided. Furthermore, if there is to be a seasonality of *x* detected, then more than 2x ( and ideally 3x) data points must be provided. For example, if the season is 12, then more than 24 data points must be provided, but ideally 36 points). For all forecasting activities it is best to provide as much historical information as possible.

#### *Too many ignored data points*

If there are too many gaps in the historical data, a forecast may not be completed successfully. Try providing more historical data and/or ignoring less by filling in the values yourself rather than relying on the engine to fill in values.

### Successful forecast

#### *Peculiar forecasted data*

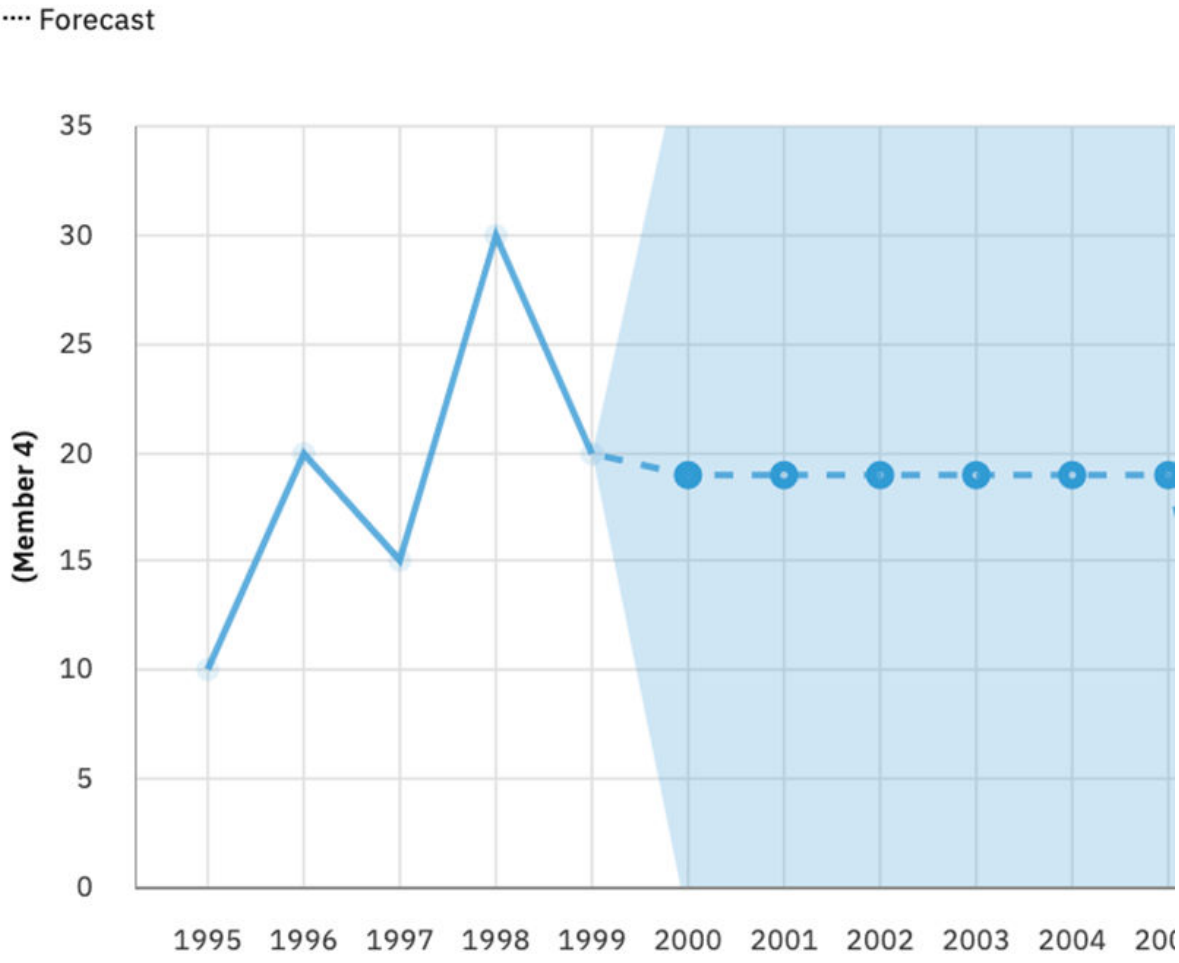
If the forecasted data seems terribly wrong, then the time axis is likely not set up properly. See the setup section “Forecasting requirements” on page 527 for details on how to set it up correctly.

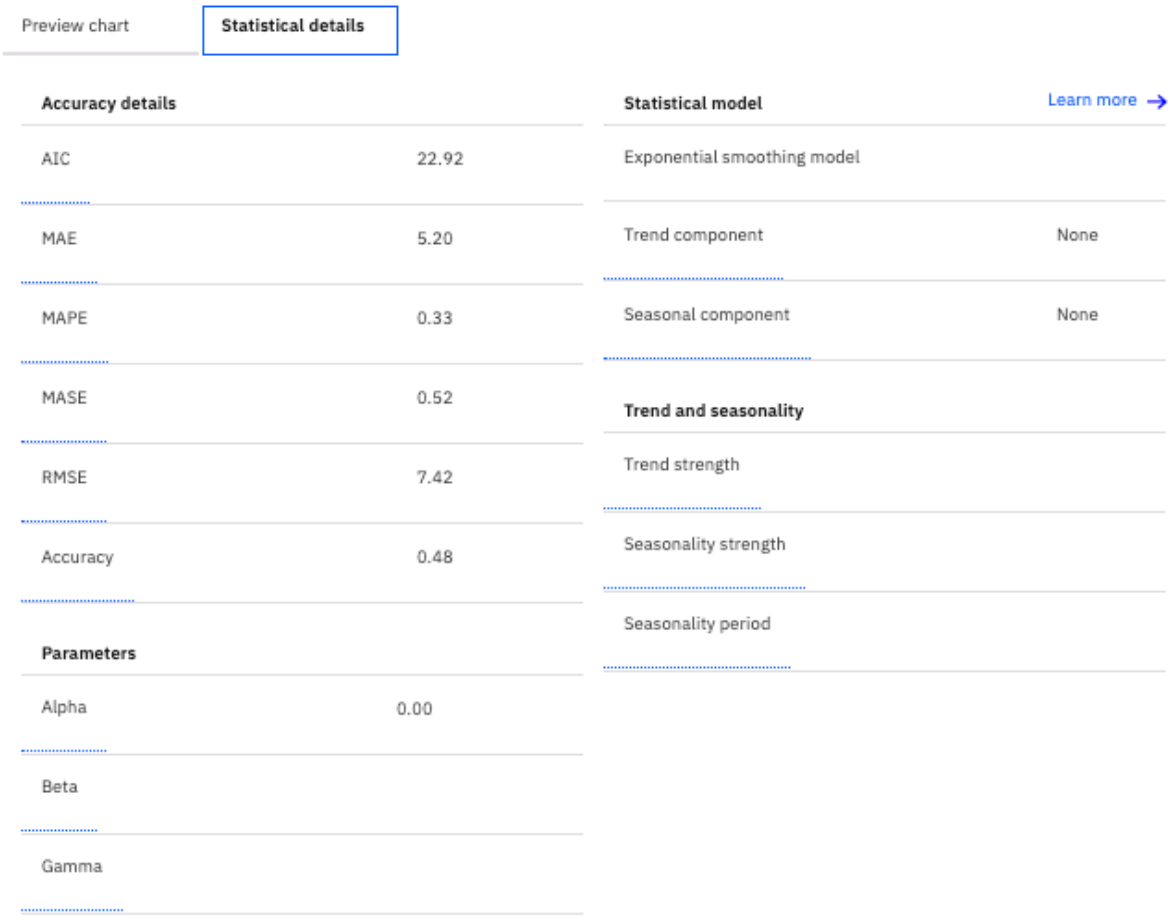
Additionally, sparse data in a cube can result in a forecast with low prediction accuracy that show exceedingly low negative values or high positive values. In this case, review the cube for sparsity and verify that the historical data is in line with predictions.

#### *Horizontal line*

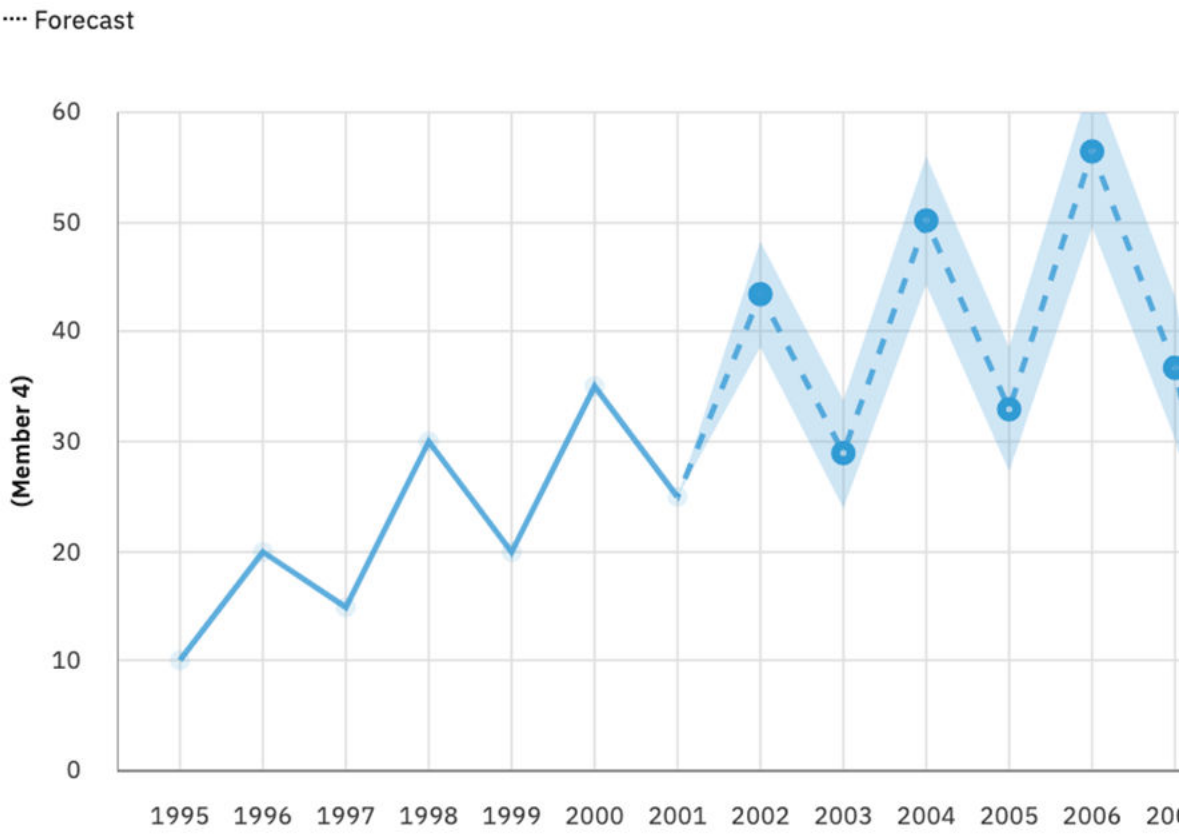
A horizontal line typically occurs when historical data is sparsely available resulting in trend or seasonality not to be detected. The following examples show where this happens and what can be done about it.

#### *Trend*


Forecast
35
30
25
20
15
10
5
0
(Member 4)
1995
1996
1997
1998
1999
2000
2001
2002
2003
2004




Not enough data is available to detect the trend that one might expect. This is confirmed by the fact that the **Trend component** that is returned (see the statistical details) is **None**. Providing more historical data allows for appropriate detection. This can also be solved by including more history. See the following example.




Preview chart | **Statistical details**

| **Accuracy details** | |
|---|---|
| AIC | 16.07 |
| MAE | 0.71 |
| MAPE | 0.03 |
| MASE | 0.07 |
| RMSE | 1.53 |
| Accuracy | 0.93 |

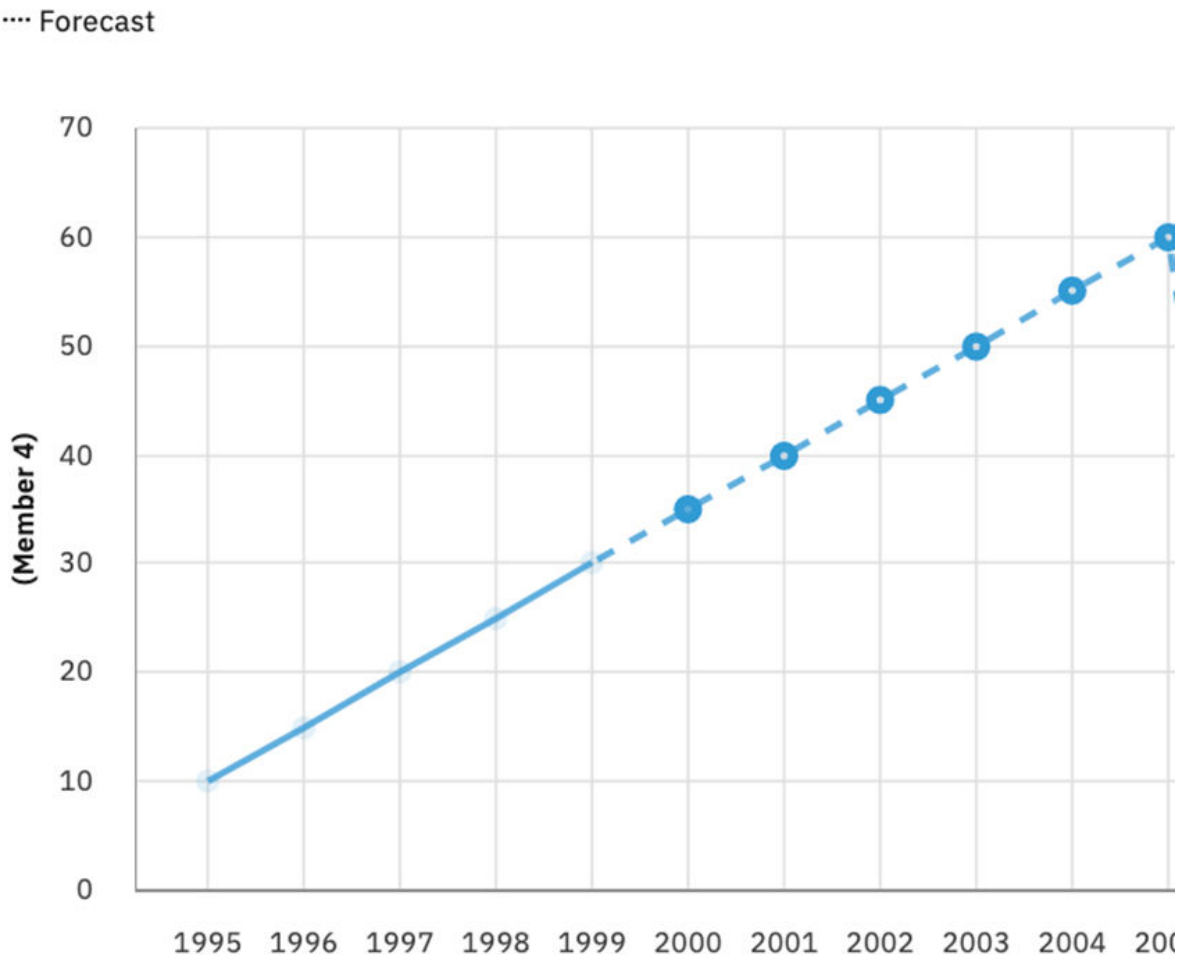
| **Parameters** | |
|---|---|
| Alpha | 0.20 |
| Beta | 0.23 |
| Gamma | 0.29 |

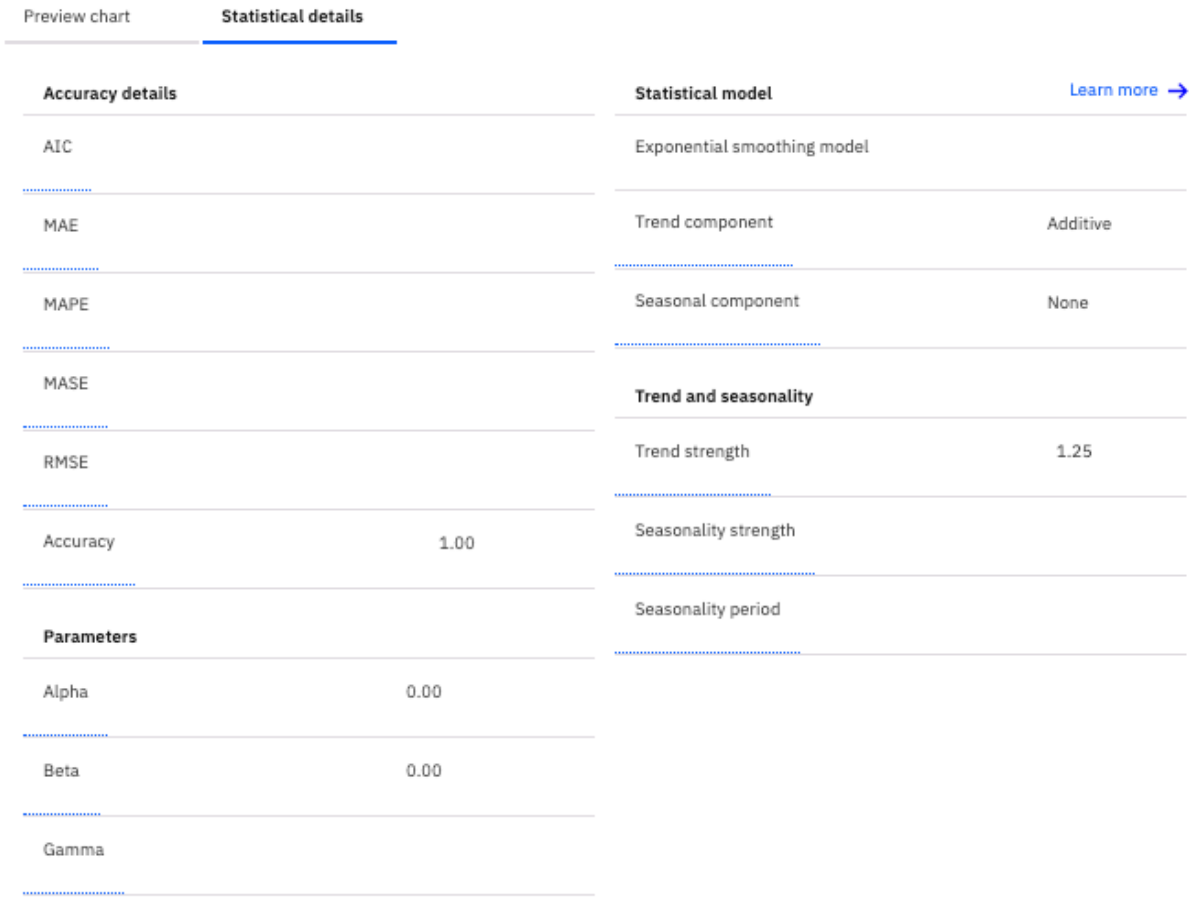
| **Statistical model** | Learn more → |
|---|---|
| Exponential smoothing model | |
| Trend component | Additive Damped |
| Seasonal component | Multiplicative |

| **Trend and seasonality** | |
|---|---|
| Trend strength | 0.22 |
| Seasonality strength | 0.50 |
| Seasonality period | 2 |

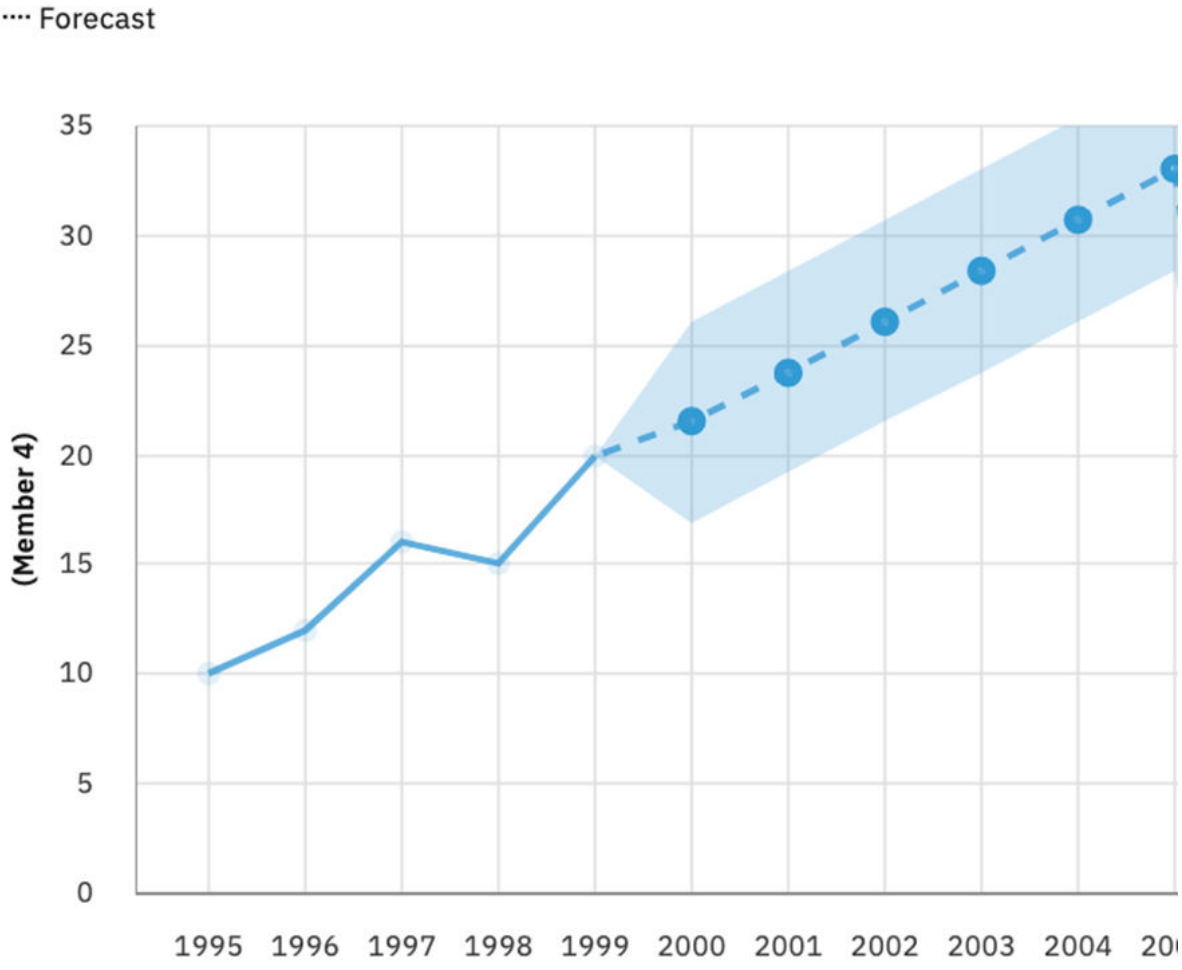
***Confidence interval envelope***

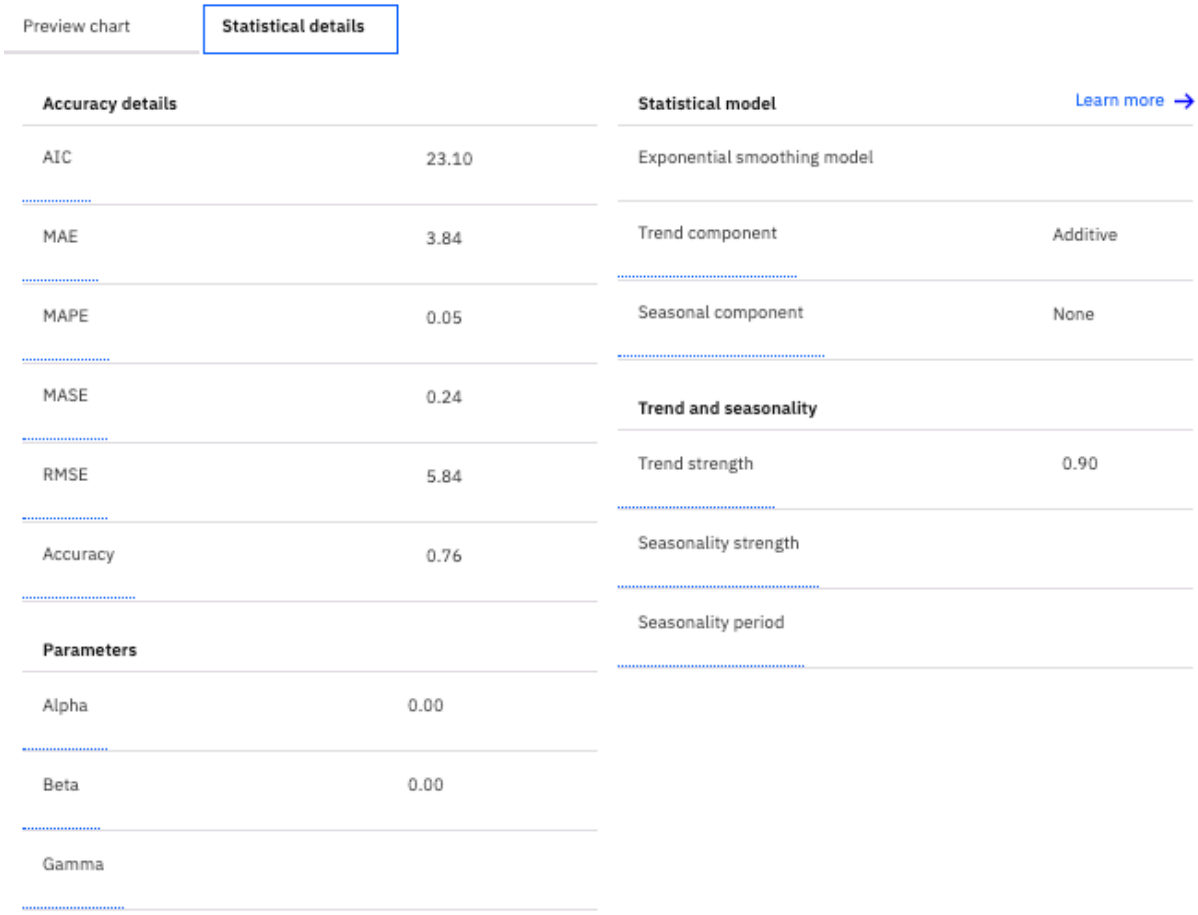
The width of the confidence envelope is based largely on how well the model matched the historical data and the confidence interval that is indicated by you (that is, 95%). What this means is that, based on the strength of match to the historical data, this is how confident the forecasted values are within this range. In the following example, with a perfect match, effectively no envelope is displayed.






However, with this example, which is given the variance of data the prediction expects continued variance along the same trend line. Trend is detected in this case.


Forecast
35
30
25
20
15
10
5
0
(Member 4)
1995
1996
1997
1998
1999
2000
2001
2002
2003
2004

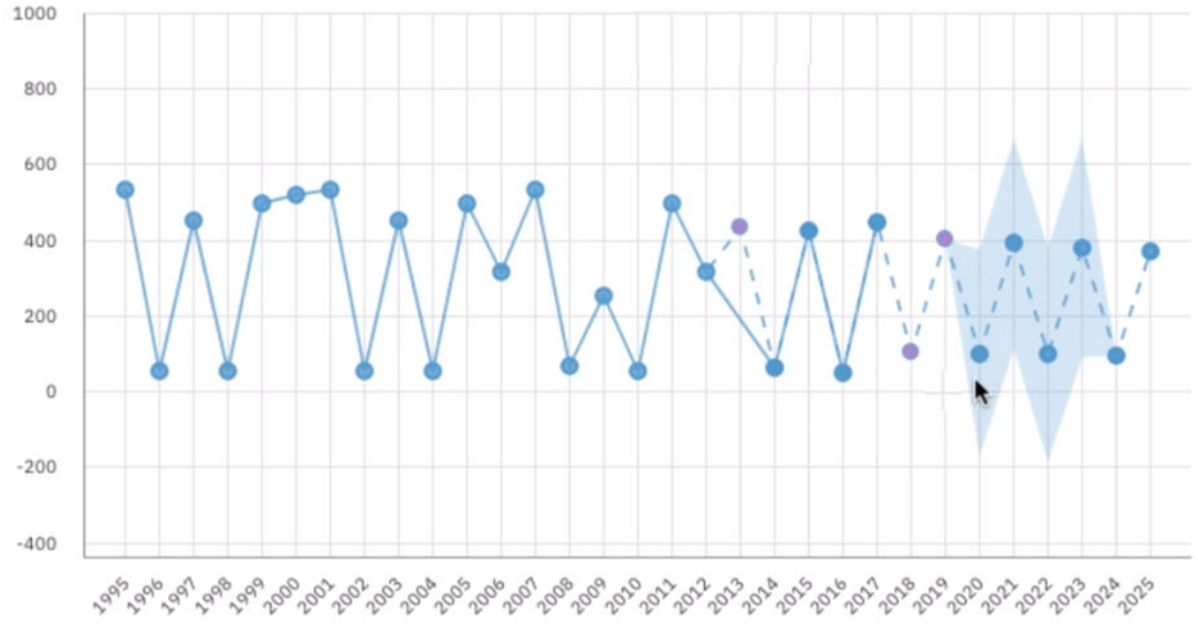



***New outliers appear after correction***

After choosing to ignore outliers, new outliers can arise during this process. This is because the act of correcting outliers changes the model that matches the historical information and confidence intervals. With a change in historical confidence intervals, new outliers might be detected.

***Confidence interval gets narrower over time***

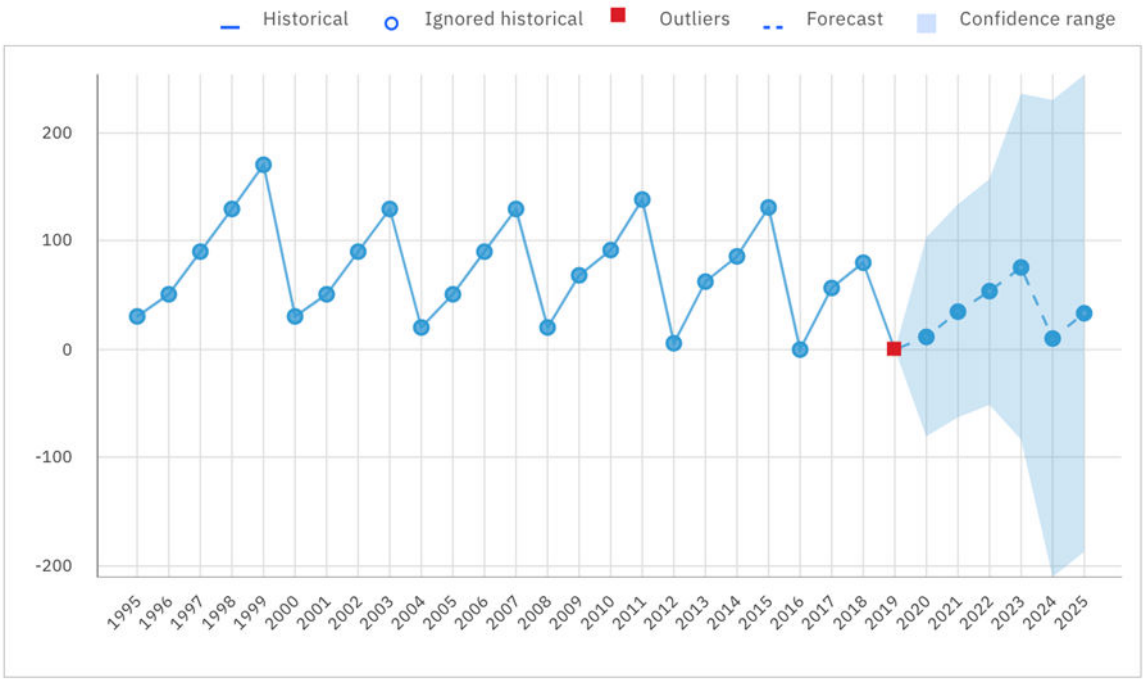
In some cases, the projected confidence interval gets narrower over time.



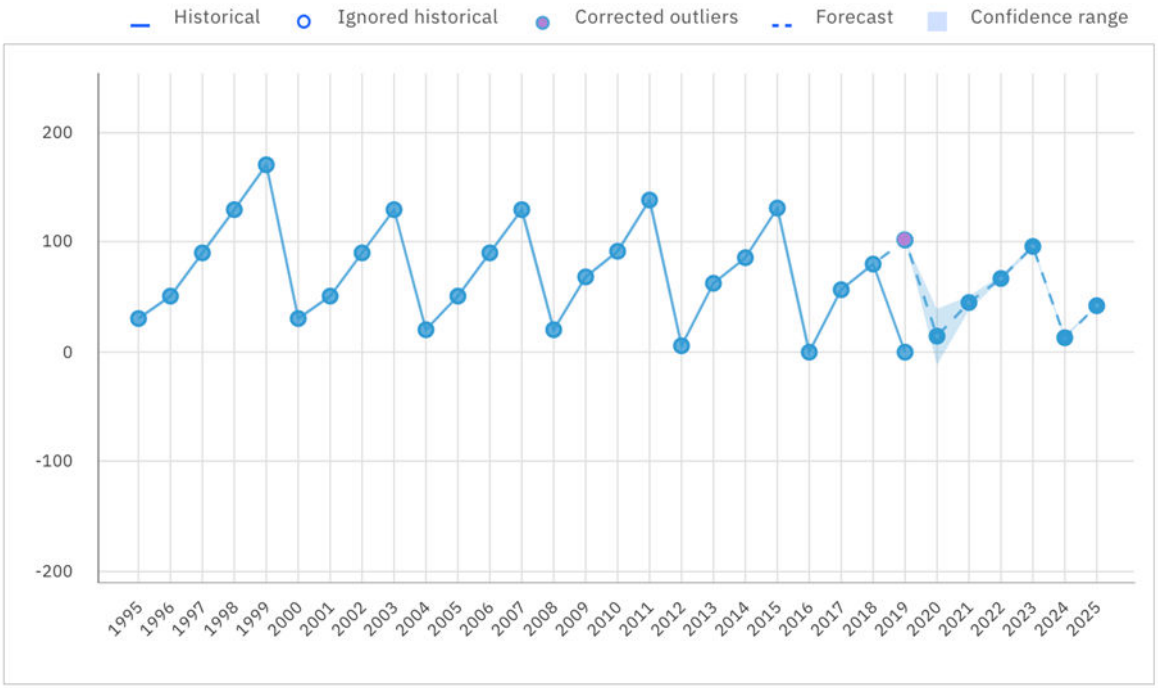
In the previous case, you can see that the confidence level doesn't exist for 2025. Intuitively we know that the future is increasingly unknown, however what is illustrated and forecasted is a result of only the known historical information.

***Graph appears disconnected***

Sometimes an outlier is detected at the very start of the forecast period, as in this example.




Correcting the outlier may cause the graph to appear disconnected, as shown here.




This is expected. The solid line represents the historical data, which ends at the corrected historical data point. Forecasted data continues from the corrected historical data point.

***Peculiar positioning of outliers***

At first glance, some outlier identifications might seem peculiar, as in this example.




You might have expected the last historical point at 2019 to be the sole outlier and none anywhere else. This would very likely have been the case if the seasonality detected was 4. However, in this case it detected a seasonality of 2. Thus, from the point where outliers were detected, everywhere it expected a cycle and there wasn't one, an outlier was positioned. The purpose of showing this is to illustrate that there are many factors that contribute to outlier detection.

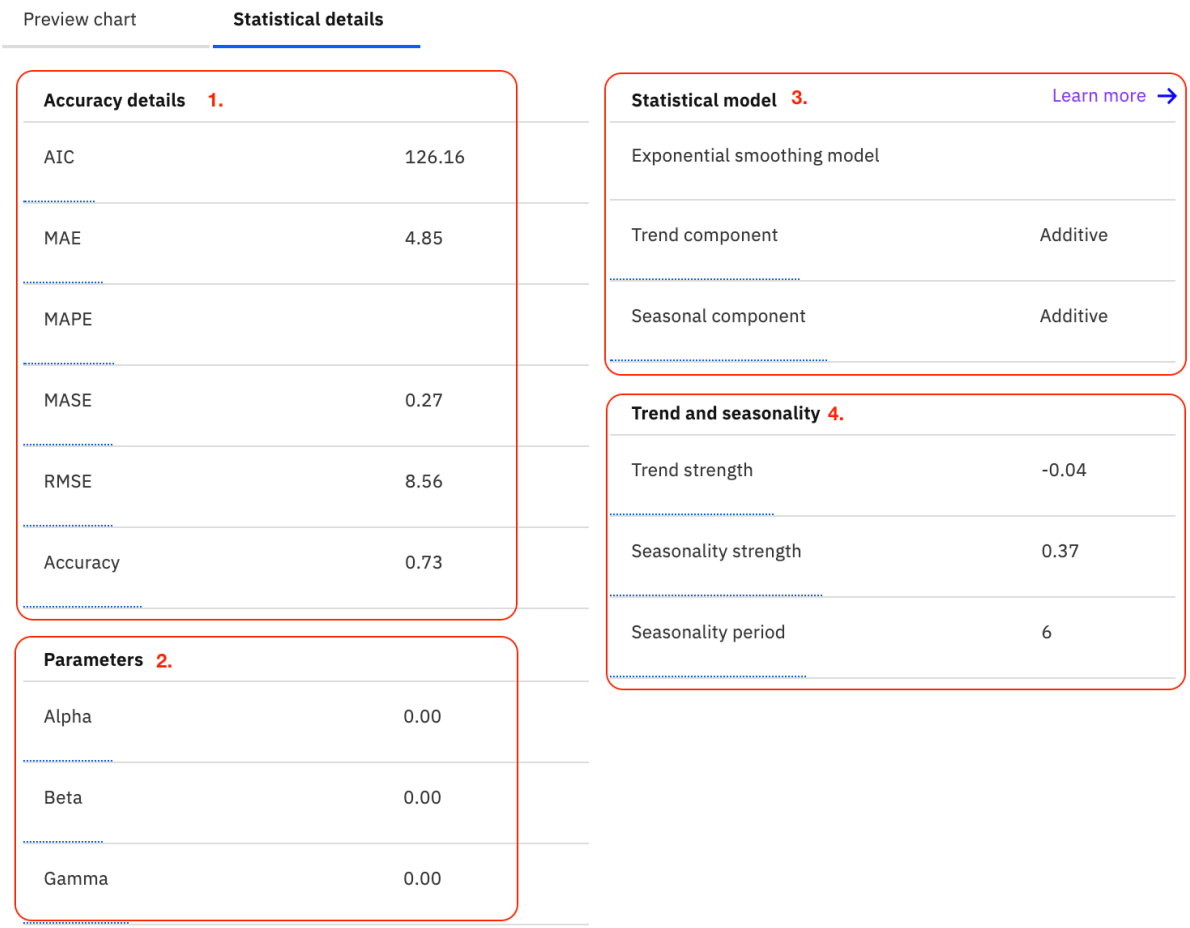
## Statistical details explained

A forecasting run generates forecasts and forecasting statistical details. Forecasting statistical details are located on the **Statistical details** tab on the **Forecasting preview**.

Review these details to gain a better understanding of your forecast.

### *Univariate statistical details*

Univariate statistical details include **Accuracy details**, **Parameters**, **Statistical model**, and **Trend and Seasonality**.




## Accuracy details (1)

Model accuracy is based on the historical data of the time series data that is used to generate the model. Accuracy details can also be used as an indicator of the forecast accuracy, but they do not carry over to future values.

**Akaike Information Criterion (AIC)**
: A model selection measure. The AIC penalizes models with many parameters, and so attempts to choose the best model with a preference towards simpler models. The AIC is the sum of the logarithm of non-adjusted MSE multiplied by the number of historical points and the number of model parameters and initial smoothing states that are multiplied by 2.

**Mean Absolute Error (MAE)**
: Computed as the average absolute difference between the values fitted by the model (one-step ahead in-sample forecast), and the observed historical data.

**Mean Absolute Percent Error (MAPE)**
: The average absolute percent difference between the values that are fitted by the model and the observed data values.

**Mean Absolute Scaled Error (MASE)**
: The error measure that is used for model accuracy. It is the MAE divided by the MAE of the naive model. The naive model is one that predicts the value at time point t as the previous historical value. Scaling by this error means that you can evaluate how good the model is compared to the naive model. If the MASE is greater than 1, then the model is worse than the naive model. The lower the MASE, the better the model is compared to the naive model.

**Root Mean Squared Error (RMSE)**

The square root of the MSE. It is on the same scale as the observed data values. Mean Squared Error (MSE): The sum of squared difference between the values that are fitted by the model and observed values that are divided by the number of historical points, minus the number of parameters in the model. The number of parameters in the model is subtracted from the number of historical points to be consistent with an unbiased model variance estimate.

**Mean Squared Error (MSE)**

The sum of squared difference between the values that are fitted by the model, and observed values that are divided by the number of historical points, minus the number of parameters in the model. The number of parameters in the model is subtracted from the number of historical points to be consistent with an unbiased model variance estimate.

## Parameters (2)

Detected Seasonal period and estimates for other parameters that are used in the selected exponential smoothing model are available.

**Alpha**

The smoothing factor for level states in the exponential smoothing model. Small values of alpha increase the amount of smoothing, that is, more history is considered when the alpha is small. Large values of alpha reduce the amount of smoothing, which means that more weight is placed on the more recent observations. When the alpha is 1, all the weight is placed on the current observation.

**Beta**

The smoothing factor for trend states in the exponential smoothing model. This parameter behaves similar to alpha, but is for trend instead of level states.

**Gamma**

The smoothing factor for seasonality states in the exponential smoothing model. Serves the similar role as alpha, but for the seasonal component of the model.

**Phi**

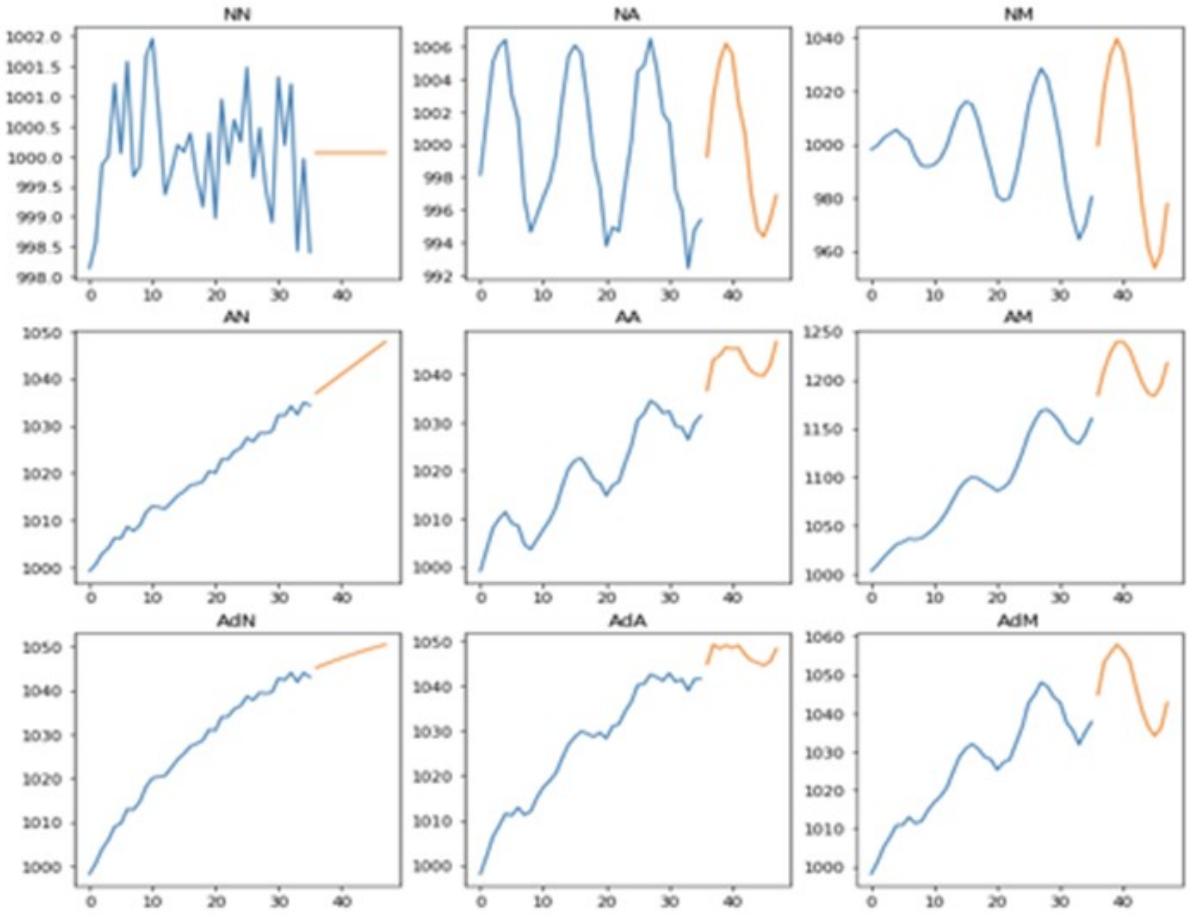
The damping coefficient in the exponential smoothing model. Long forecasts can lead to unrealistic results, and it is useful to have a damping factor to dampen the trend over time and produce more conservative forecasts.

## Statistical model (3)

Forecasting uses exponential smoothing. Every exponential smoothing model that is fit contains a level component. However, trend and seasonal components are not always present. The following table shows the different types of exponential smoothing models that are fit and accompanying time series data examples for each model type.

| **Trend component** | **Seasonal component** | | |
|---|---|---|---|
| | None | Additive | Multiplicative |
| None | Simple exponential smoothing | Simple exponential smoothing | |
| Additive | Holt's linear method | Additive Holt-Winters' method | Multiplicative Holt-Winters' method |
| Additive damped | Additive damped trend method | Additive Holt-Winters' method with damped trend | Multiplicative Holt-Winters' method with damped trend |

The following diagram illustrates what the detected model might be with the given historical data (blue) and the resulting forecasting data (yellow).




Time series data that contains a trend, for example, slope, can be modeled with a trend component. The trend can be damped or non-damped. Damped trend eventually levels off into a flat line, which leads to a more conservative fit while undamped would continue to grow without bounds.

Time series data that contains regularly repeating cycles, can be modeled with a seasonal component. The seasonality can be additive or multiplicative. An example of multiplicative seasonality is an amplitude that is constantly growing or shrinking with time. If the seasonal pattern is constant, then additive seasonality is used.

**Trend component**

Identifies the trend component. Supported types include: None (trend not detected), Additive and Additive damped.

**Seasonality component**

Identifies the seasonality component. Supported types include: None (seasonality not detected), Additive and Multiplicative.

## Trend and seasonality (4)

**Trend strength**

Compare the original model, M, and the same model with the trend component removed. The trend strength of M is the difference in accuracy between model M and model M with the trend component removed.

**Seasonality strength**

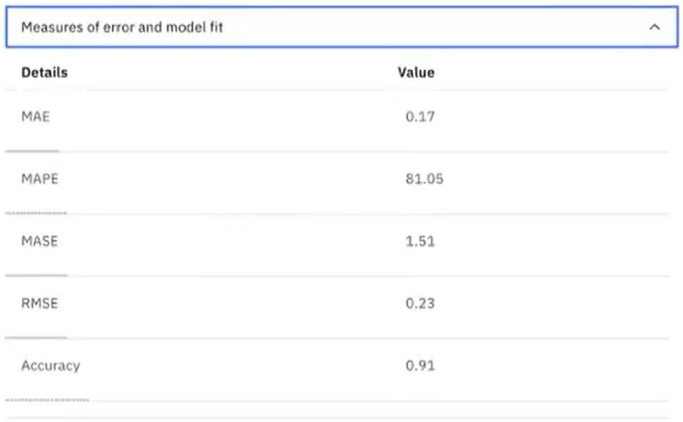
Compare the original model, M, and the same model with the seasonal component removed. The seasonality strength of M is the difference in accuracy between model M and model M with the seasonal component removed.

**Seasonality period**

The number of contiguous items detected per cycle. For example, if the finest granularity of the time series is months, then a number of 12 here would represent an expectation that a repeating pattern yearly exists.

### *Multivariate statistical details*

Multivariate statistical details describe **Measures of error and model fit**.




**Mean Absolute Error (MAE)**

Computed as the average absolute difference between the values fitted by the model (one-step ahead in-sample forecast), and the observed historical data.

**Mean Absolute Percent Error (MAPE)**

The average absolute percent difference between the values that are fitted by the model and the observed data values.

**Mean Absolute Scaled Error (MASE)**

The error measure that is used for model accuracy. It is the MAE divided by the MAE of the naive model. The naive model is one that predicts the value at time point t as the previous historical value. Scaling by this error means that you can evaluate how good the model is compared to the naive model. If the MASE is greater than 1, then the model is worse than the naive model. The lower the MASE, the better the model is compared to the naive model.

**Root Mean Squared Error (RMSE)**

The square root of the MSE. It is on the same scale as the observed data values. Mean Squared Error (MSE): The sum of squared difference between the values that are fitted by the model and observed values that are divided by the number of historical points, minus the number of parameters in the model. The number of parameters in the model is subtracted from the number of historical points to be consistent with an unbiased model variance estimate.

**Accuracy**

The estimated predictive accuracy of the forecast.

## Forecasting error messages

You might receive error or informational messages about problems that occur while forecasting. Use the information here to help resolve problems you encounter. If you receive an error not listed here, try to perform the action that caused the error again and contact IBM Support if the error persists.

**Disabling zero suppression may clear this error**

This message generally accompanies other messages. Disable zero suppression on the view and try again. For more information, see Forecasting setup - Zero suppression.

**Forecasting did not succeed for one or more series because the data has too many missing values (more than 33%)**

The member to be forecast contains too many blank or missing values.

**Forecasting did not succeed for one or more series due to an insufficient number of historical data points (at least 5 non-ignored points are required)**

After ignoring data points, a minimum of five data points is required for a valid univariate forecast. Generally, the historical duration should be at least 3x larger than the forecast period.

**Forecasting did not succeed for one or more series due to an insufficient number of historical data points (at least 16 non-ignored points are required)**

Too few historical points for multivariate forecasting. Minimum number of data points required for multivariate VAR is 16 while minimum data points required for multivariate ARIMAX is 63.

**Forecasting succeeded, but some missing values were found in the data and imputed**

The accuracy of the forecast was less than optimal because there are too many missing values.

**Ignore period cannot be greater than start period**

The ignore period (date) cannot be within the forecast period. Change the ignore or forecast periods to prevent overlap.

**In order to preview remove the sandbox dimension from the context bar**

Move the sandbox dimension out of the context area. Click the **Action** icon ⋮ for the sandbox dimension, then click **Hierarchy** > **Remove**. Then, try the preview again.

**Invalid start time or end time**

Planning Analytics Workspace validated data entries, but the start or end time is invalid. Select a valid time period using the selection UI.

**Model could not be trained and evaluated, as the data has too many missing values (0's) in continuity**

There are too many zeroes in the data, which inhibits proper training and evaluation of at least one of the forecast models.

**One of the cells used as a reference for spreading has a value of 0**

A spreading reference cell cannot be zero. Select a non-zero reference cell.

**Please select a leaf-level member**

This message indicates that a leaf level (non-consolidated) member was not selected. Using the member picker, change the selection to a leaf-level member (for spreading reference).

**Something went wrong while generating the forecast for one or more series**

This is a general message for any unsuccessful prediction. Please try again and if the issue persists contact IBM Support.

**The forecast comments cannot exceed {#}**

There is a limit to applying comments to selected members. You cannot apply comments to more than 25 members.

**The forecast data does not match the cells to be written to**

This error occurs when you try to forecast more values (time periods) than exist in the underlying cube. Adjust the forecast period or request that additional time periods be added to the cube.

**The predictive forecast for {cubeName} did not complete successfully.\n%{error}**

This is a general message for any unsuccessful prediction. Please try again and if the issue persists contact IBM Support.

**Too few historical values**

Increase the historical duration. Generally, the historical duration should be at least 3x larger than the forecast period.

**Too many missing points**

Forecasting is not possible for elements with too many blank or missing values.

**Unable to forecast on control cube**

The forecasting feature does not support control cubes.

**Unable to forecast on locked cube**

You cannot forecast values in a locked cube. Unlock the cube if you want to proceed with forecasting.

**Unable to forecast on the rule derived cell**

Predictive forecasting do not support writing a forecast value to a rule-derived cell. Select another save location (in Advanced tab) that is not a rule-derived cell.

**Unable to update cell data because of Consolidation Hold**
Certain combinations of holds are not possible with value spreading. Release holds to correct.

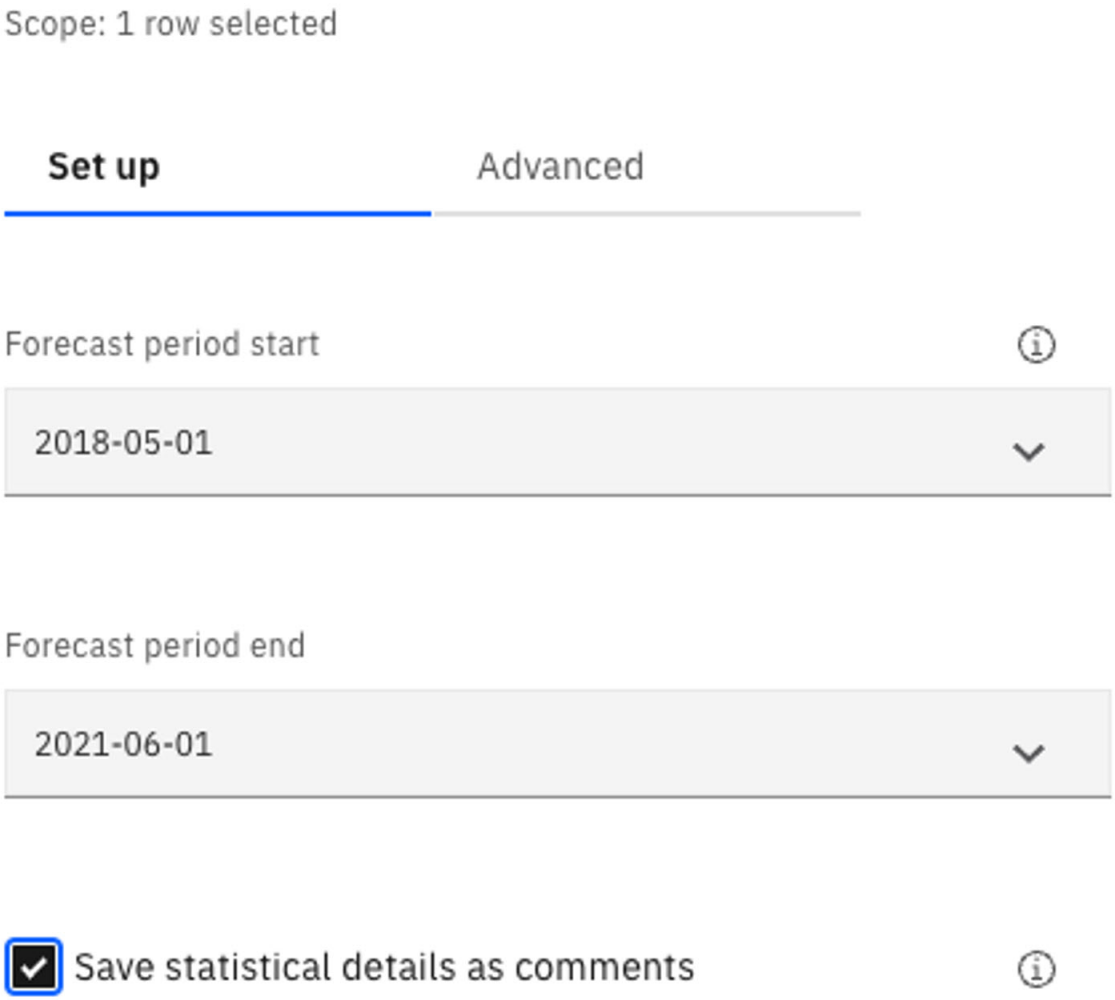
**Variable(s) future values can not be considered due to insufficient historical datapoints (minimum 63)**
There are too few time points to run ARIMAX exogenous case.

# Forecasting on actuals

When the forecast options are set up and you previewed the forecast, you can forecast on the actuals.

## Before you begin

By clicking **Forecast**, you change the TM1 data. The settings in the **Forecast** dialog box are tied to the crosstab used for the process. To share the forecasting parameters with other users, save the book. Different views even the ones that are created from the same TM1 cube do not share parameters.



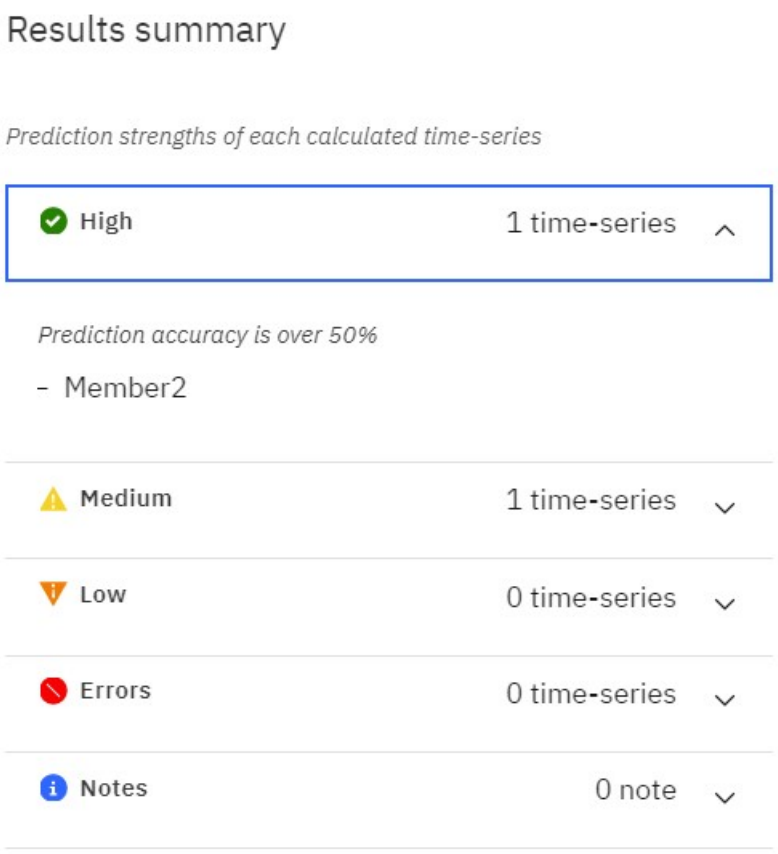

## Procedure

1. On the **Forecast** dialog box, click **Forecast**.

   It takes some time for the forecast to complete. When completed, a message is displayed that the AI driven forecast completed successfully.

   The data in the crosstab refreshes.

The **Result summary** and the **Details** display information on the quality of the forecast. The details specify which time series landed on each quality hierarchy (High, Medium, Low, Errors). The accuracy measure from the forecasting engine defines the hierarchy.




2. The **Statistical details** tab, shows all the details for the forecast.
3. If you do not want the forecasted date to be displayed in cells in the crosstab, use the option **Save statistical data as comments**. For each time-series forecast row, statistical details are saved as a comment on the first forecast cell. Available for up to 25 rows.

# Forecasting models

Exponential smoothing models are a popular class of time series models.

Exponential smoothing models are applicable to a single set of values that are recorded over equal time increments only. However, they support data properties that are frequently found in business applications such as trend, seasonality, and time dependence. All specified model features are estimated based on available observed data. An estimated model can then be used to forecast future values and provide upper and lower confidence bounds for the forecast values.

Each model type is suited for modeling a different combination of properties that are found in the data. The model type that can provide the best match to the observed data is selected for modeling the observed data and is used to forecast any future values.

## Model estimation algorithms

Models are specified by the smoothing equations that include the model parameters and initial smoothing states. Model parameters are estimated with values that minimize the model error.

## Smoothing equations

Exponential smoothing models derive their name from the smoothing equations that specify the model. They provide formulas for computing smoothing states for each observed point by using the current observed value and the previous smoothing states. Smoothing equations provide weighted averages of the current value and the previous states in the time series. Weight for the current value or state is given by a model parameter between 0 and 1, while the weights for previous values are exponentially decreasing.

## Level smoothing equations

All model types compute a level state for each time series point by using the corresponding level smoothing equation. Level states for the model without trend and seasonal components are computed as the weighted average of the time series value at the current point and the level state at the previous point. The weight that is associated with the current value is a parameter, **alpha**, with its value restricted between 0 and 1. For other models, previous trend and seasonal states are also included in the level smoothing equation.

## Trend smoothing equations

Model types with additive or damped additive trend compute a trend state for each time series point by using the corresponding trend smoothing equation. The trend state for the current point is based on the difference of level states at the current and the previous point, and on the trend state at the previous point. The weight that is associated with the difference of level states at the current and previous point is a parameter that is named **beta** with its value restricted between 0 and 1. An extra parameter, **phi**, is added to the damped trend smoothing equations. **Phi** multiplies the trend state contribution from the preceding point and its value is also restricted between 0 and 1. The purpose of this parameter is to estimate the degree of trend damping from one point to the next.

## Seasonal smoothing equations

Model types that support additive or multiplicative seasonality compute a seasonal state for each time series point. The seasonal states are computed by using seasonal smoothing equations. The seasonal state for the current point includes the difference of the time series value and the current level state for additive seasonality or ratio of the two same values for multiplicative seasonality. The weight that is associated with this term is a parameter, **gamma**, with its value restricted between 0 and 1. The rest of the contribution comes from the corresponding seasonal state in the previous seasonal period. Notice that the seasonal period has a fixed length, and while the seasonal state can change for each point, only matching seasonal indices from different periods are considered together in the seasonal smoothing equations.

## Initial smoothing states

Values must be specified for level, trend, and seasonality states for points that precede the time series. The values are needed for the smoothing equations. To compute the various states at the first point of the time series requires state values at the corresponding previous points.

## Model parameters

Each smoothing equation uses corresponding model parameters:

**alpha**
: Controls the level states.

**beta**
: Controls the trend states.

**gamma**
: Controls seasonal indices across seasonal periods.

**phi**
An extra parameter that is used for specifying the damped trend.

All four parameters have values between 0 and 1. Higher values of **alpha**, **beta**, and **gamma** mean that more recent observations have higher weight, while lower values mean higher weights for older observations. A higher value of **phi** corresponds to a higher degree of dampening the forecast trend.

### Model estimation

Model parameters in the smoothing equations are estimated based on the time series data. Parameters cannot be estimated directly by using a formula. They are estimated by an iterative process that searches for parameter values that minimize the model error. The model error is computed as Mean Absolute Scaled Error (MASE). The iterations stop when no further reduction in the model error can be achieved. Corresponding parameter values together with the initial smoothing states fully specify the estimated model. They are used to compute the model states for all other data points and generate the model forecasts by using a corresponding forecast equation.

## Forecasting algorithms

A number of algorithms are used in forecasting.

### One-step ahead

Every model supports one-step ahead forecasts based on the corresponding forecast equation. One-step ahead forecasts are needed to compute model errors during the model estimation process.

One-step ahead forecasts are computed sequentially for each data point by using computed level and trend states for the current point, and seasonal states for the last seasonal period.

Forecast error is computed by subtracting forecast value (estimated at the previous point) from the observed value at the current point. Overall model error, which is used for estimating the model, is computed as an average value of absolute squared forecast errors. Smaller errors correspond to a better model fit. Accuracy measures displayed in **Statistical details** provide several model summaries of the one-step ahead forecast errors.

### k-step ahead

k-step ahead forecasts are used to make predictions for any number of future values following the observed time series data. They are based on the same forecast equations as the one-step ahead forecasts for the specified model.

The number of forecast values that are generated is 20% of the length of historical data series by default. You can specify an exact number of values to be forecast in the **Forecast** dialog box. Any missing values at the end of a particular series will also be forecast, but they will not count towards the specified number of forecast periods.

### Confidence bounds

Confidence bounds provide the level of uncertainty that is associated with each forecast value. The bounds typically become wider further into the future, because more distant forecasts are less reliable. Confidence bounds provide relevant insights into the future behavior of the observed time series.

Computation of confidence bounds is based on the overall variance of forecast errors that are estimated on the observed data and a factor that depends on the specified model and on the number of steps from the last observed point.

### Automated model selection in forecasts

Multiple model types are used to create candidate models for each time series in a forecast. All available model types are normally used, except when a seasonal component is absent. There are only three model types available that do not account for seasonality in the data.

The default value of **Auto** for the **Seasonal period** option detects seasonal period length through spectral analysis. A model is then trained using the period detected through spectral analysis.

Multiple models are compared by using a model error and the number of model parameters. For example, when model errors are equal for two models, the model with fewer parameters is preferred. The latter model provides a more condensed representation of the observed data and also tends to generate more reliable forecasts.

## Forecasting tutorial

This tutorial guides you through the creation of a univariate forecast in IBM Planning Analytics Workspace.

The creation of a multivariate forecast is similar, but you would specify one or two forecast variables for a multivariate forecast.

### Forecasting tutorial: preparing data

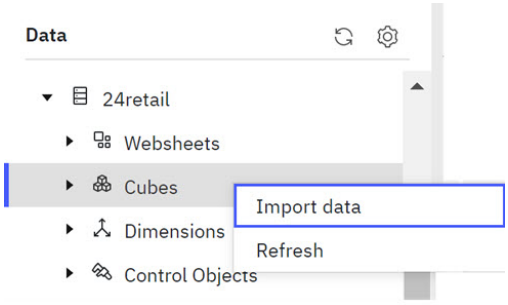
For the forecast tutorial, you download a sample file and set up the sample data in IBM Planning Analytics Workspace.

#### Before you begin

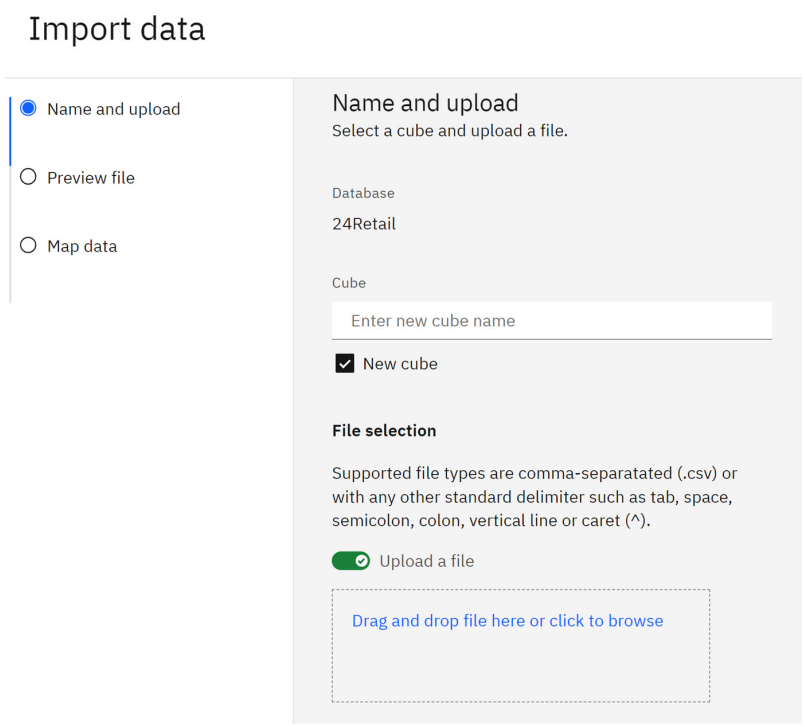
This tutorial is meant to be followed in sequential order. Each part assumes that the previous part was completed.

#### Procedure

1. Download the sample files `pa_forecasting_tutorial.csv` and `pa_data_spreading_demo.csv` from https://www.ibm.com/support/pages/node/6334299. You'll use `pa_forecasting_tutorial.csv` immediately and use `pa_data_spreading_demo.csv` in a later tutorial.
2. Log in as an administrator or modeler.
3. Create a Planning Analytics Workspace book.
4. Expand the **24Retail** database on the **Data** tree.
5. Right-click **Cubes** then click **Import data**.

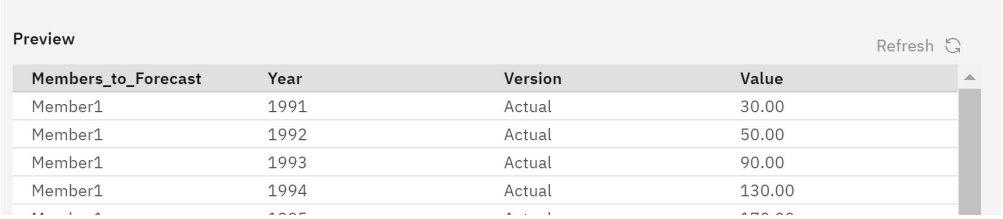



6. On the **Import data** window, click **New cube**.




7. Name the cube `Tutorial_<your initials>`, using your initials as a suffix. You'll return to this cube in later parts of the tutorial, so it's important to use this name.
8. Upload the `pa_forecasting_tutorial.csv`file, then click **Next**.
9. Examine the preview of the `pa_forecasting_tutorial.csv` file, then click **Next**.

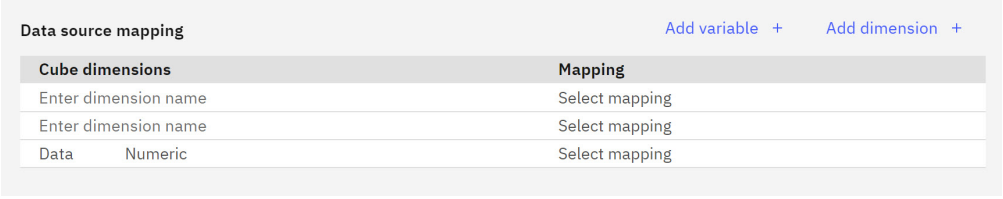
   The file contains three columns that are used to create dimensions and one column that contains numeric values.




| Members_to_Forecast | Year | Version | Value |
|---|---|---|---|
| Member1 | 1991 | Actual | 30.00 |
| Member1 | 1992 | Actual | 50.00 |
| Member1 | 1993 | Actual | 90.00 |
| Member1 | 1994 | Actual | 130.00 |

10. On the **Map data** panel, click **Leaf only**.
11. Click **Overwrite existing values**.
12. Click **Save process as**, then enter `Import_<your initials>` as the process name.

    Now it's time to map the columns in the `pa_forecasting_tutorial.csv` file to Planning Analytics dimensions. The mapping is done in the **Data source mapping** section of the **Map data** panel.




| Cube dimensions | Mapping |
|---|---|
| Enter dimension name | Select mapping |
| Enter dimension name | Select mapping |
| Data Numeric | Select mapping |

    a) Click the first **Enter dimension name** cell and enter `Members_<your initials>`.
    b) Click the corresponding **Select mapping cell**, then click **Members_to_Forecast**.
    c) Click the next **Enter dimension name** cell and enter `Years_<your initials>`.
    d) Click the corresponding **Select mapping cell**, then click **Year**.

The **Data source mapping** section contains place holders for two cube dimension, but the `pa_forecasting_tutorial.csv` source file includes columns that map to three dimensions. You must add and map a third dimension.

e) Click **Add dimension**.

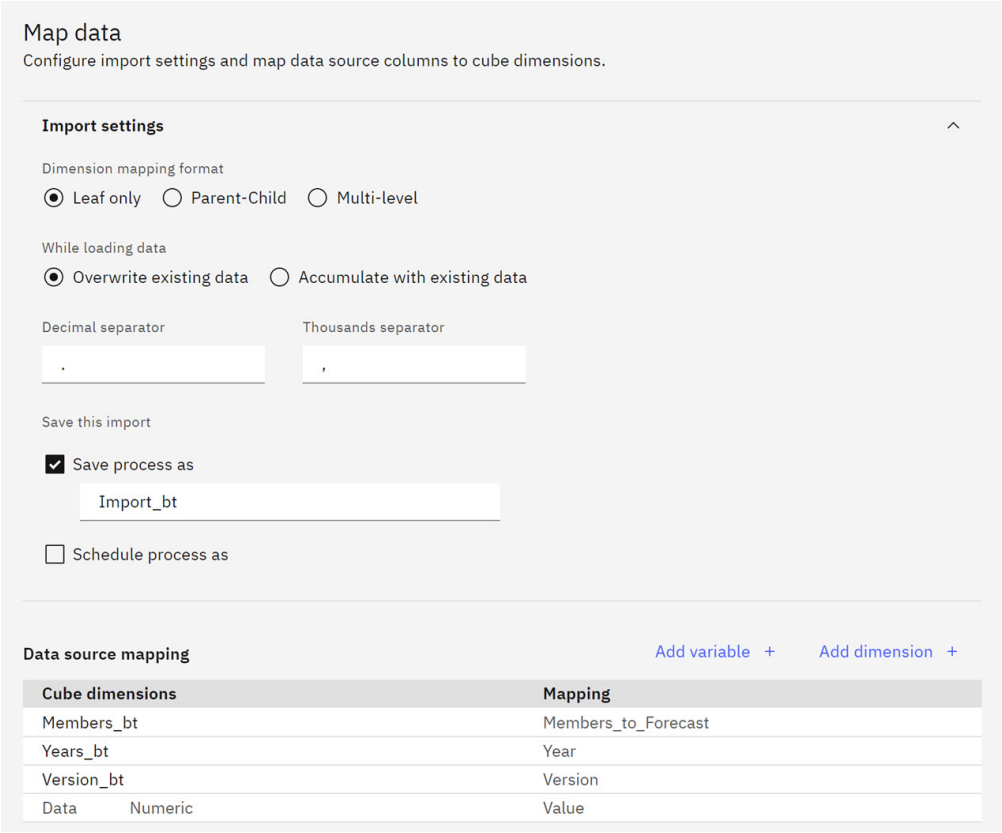
f) Click the new **Enter dimension name** cell and enter `Version_<your initials>`.

g) Click the corresponding **Select mapping cell**, then click **Version**.

The last column in the data source is correctly identified as Data of Numeric type, but it isn't yet mapped.

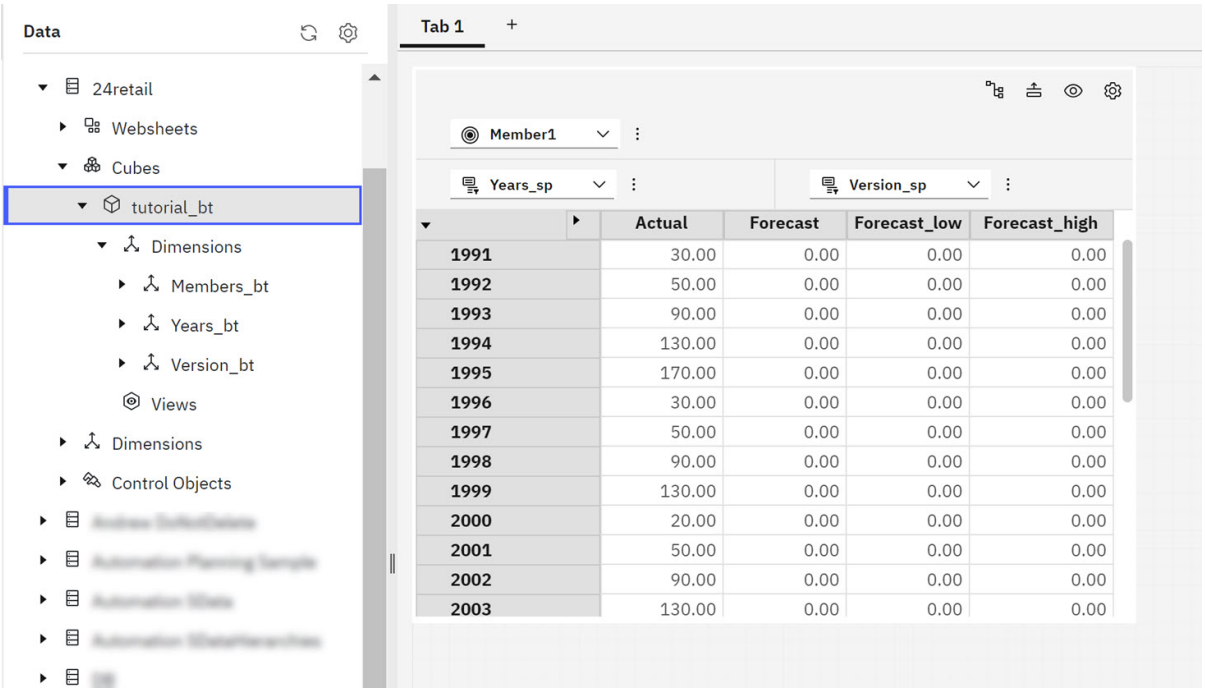
h) Click the **Select mapping cell** that corresponds to Data, then click **Value**.

The **Map data** panel now looks like this.




13. Click **Import**.
14. The **File import results** dialog reports the processing of the data source. Click **OK** to close the report.

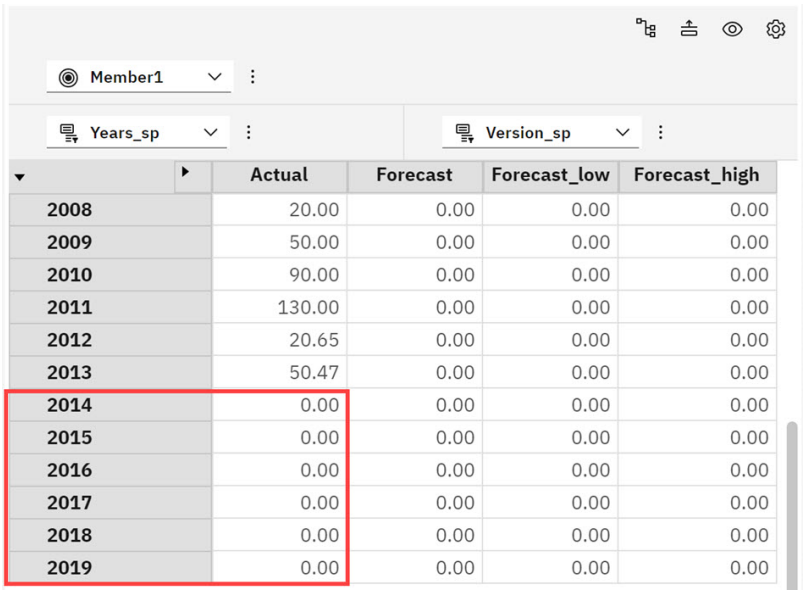
The tutorial cube is created and inserted into the book. You can click the **Show cube in data tree** icon to see the newly created cube and its dimension in the Data tree.




The sample cube has four members in the Version dimension:

| Member | Purpose |
|---|---|
| **Actuals** | Holds the user data. |
| **Forecast** | Typically used to store forecast data so that Actuals values remain untouched. The Forecast member is also used to evaluate the quality of a forecast vs the real data. |
| **Forecast_high** | Similar to Forecast, but used to store the forecasted data on the higher bound of the confidence interval. |
| **Forecast_low** | Similar to Forecast but used to store the forecasted data on the lower bound of the confidence interval. |

If you examine the new cube, you can see that there are Actual data values for the years 1991 - 2013, but no values for 2014 - 2019. This tutorial generates a forecast for the years 2014 - 2019.




| | Actual | Forecast | Forecast_low | Forecast_high |
|---|---|---|---|---|
| 2008 | 20.00 | 0.00 | 0.00 | 0.00 |
| 2009 | 50.00 | 0.00 | 0.00 | 0.00 |
| 2010 | 90.00 | 0.00 | 0.00 | 0.00 |
| 2011 | 130.00 | 0.00 | 0.00 | 0.00 |
| 2012 | 20.65 | 0.00 | 0.00 | 0.00 |
| 2013 | 50.47 | 0.00 | 0.00 | 0.00 |
| 2014 | 0.00 | 0.00 | 0.00 | 0.00 |
| 2015 | 0.00 | 0.00 | 0.00 | 0.00 |
| 2016 | 0.00 | 0.00 | 0.00 | 0.00 |
| 2017 | 0.00 | 0.00 | 0.00 | 0.00 |
| 2018 | 0.00 | 0.00 | 0.00 | 0.00 |
| 2019 | 0.00 | 0.00 | 0.00 | 0.00 |

## Forecasting tutorial: previewing results

In this part of the tutorial, you preview the results of a forecasting process. The preview operation does not modify data in your Planning Analytics database.

### Before you begin

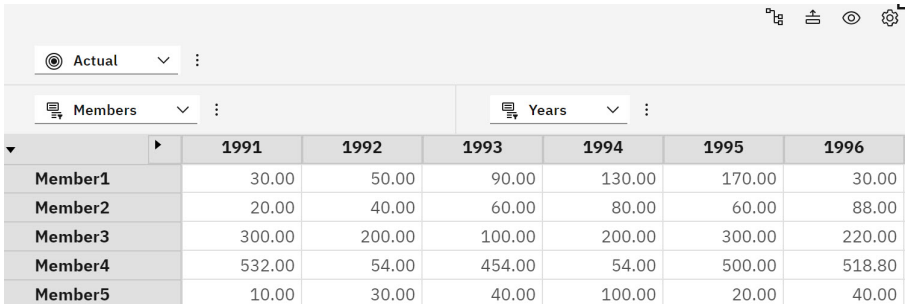
This tutorial is meant to be followed in sequential order. Each part assumes that the previous part was completed.

**Note:** Forecasting requires that the time dimension to be placed on the column in a view. The algorithm assumes that the time dimension has no gaps. Forecasting also supports hierarchical time dimensions and nested dimensions as time (that is year, quarter, and month dimensions). Exploration views with dimensions that do not have a time dimension on the column can be forecasted, but the process might fail or the results can be unreliable.

### Procedure

1. On the Planning Analytics Workspace Home menu, click **New** > **Book**.
2. Click to reveal the Data tree.
3. Expand the **24retail** database.
4. Expand the **Cubes** node under the 24retail database.
5. Right-click the `Tutorial_<your initials>` cube you created in “Forecasting tutorial: preparing data” on page 570, then click **Add new view**.
6. Use drag and drop to move the dimensions on the view:
   - Years on the columns
   - Members on the rows
   - Version on the context area, with the Actual member displayed

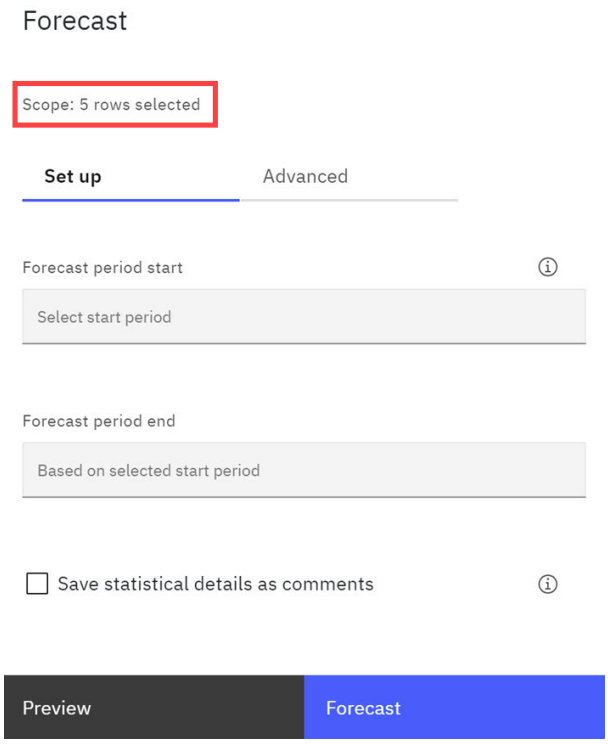
   The view should look like this:




| | 1991 | 1992 | 1993 | 1994 | 1995 | 1996 |
|---|---|---|---|---|---|---|
| Member1 | 30.00 | 50.00 | 90.00 | 130.00 | 170.00 | 30.00 |
| Member2 | 20.00 | 40.00 | 60.00 | 80.00 | 60.00 | 88.00 |
| Member3 | 300.00 | 200.00 | 100.00 | 200.00 | 300.00 | 220.00 |
| Member4 | 532.00 | 54.00 | 454.00 | 54.00 | 500.00 | 518.80 |
| Member5 | 10.00 | 30.00 | 40.00 | 100.00 | 20.00 | 40.00 |

7. Select all five members on the row dimension (Member1 - Member5), then click the **Forecast** icon on the Exploration toolbar.

   The **Forecast** panel opens. Use this panel to set parameters for your forecast. The Scope of the forecast is automatically set to the five members you selected on the row dimension.




   As you recall from preparing the cube for this tutorial, actual values exist for all years up to 2013. The forecast should start at the first year for which values do not exist, 2014.

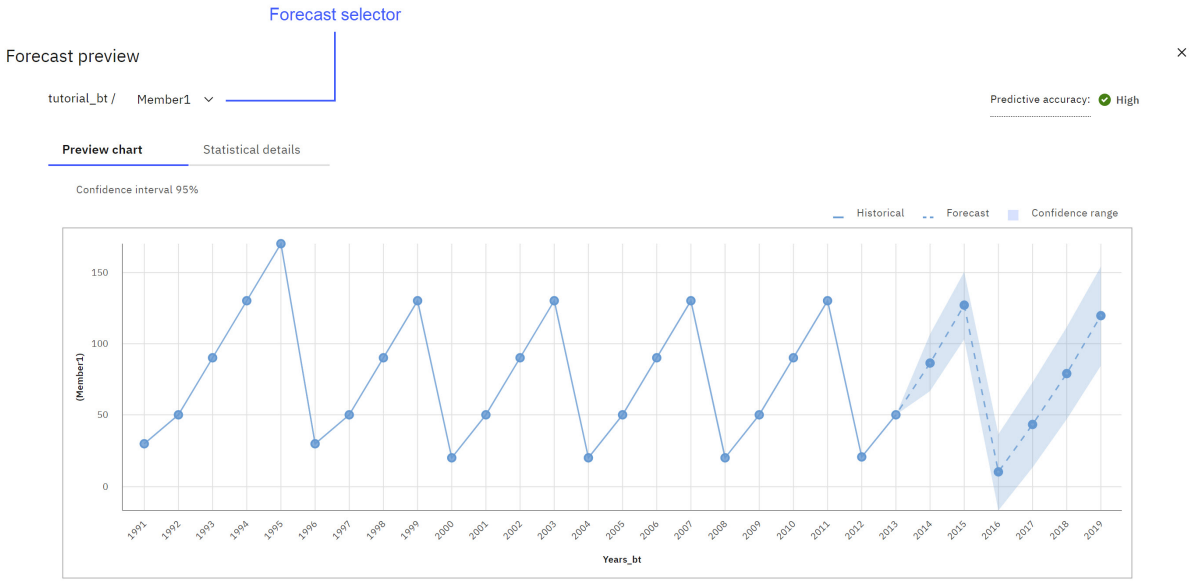



| | 2012 | 2013 | 2014 | 2015 | 2016 | 2017 | 2018 | 2019 |
|---|---|---|---|---|---|---|---|---|
| Member1 | 20.65 | 50.47 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member2 | 280.00 | 292.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member3 | 220.00 | 220.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member4 | 322.45 | 322.45 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member5 | 62.15 | 62.15 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

8. Click **Select start period**, then click **2014** to set the Forecast start period.

   The Forecast end period is automatically set to the last member in the dimension that is used to define the forecast period, in this case 2019.
9. Enable the **Save statistical details as comments** option.
10. Click **Preview**.

The preview shows a forecast chart for the first member in the forecast; in this case it's named Member1. The chart includes forecast values for the years 2014 through 2019




You can click the **Forecast selector** to preview a forecast for a different member.

11. Click **Statistical details** to review the details of the accuracy and statistical model for the preview.

    You can hover on any of the measures or parameters in the **Accuracy** section to view full definitions. For more information about the statistical model, see "Statistical details explained" on page 560.

## Forecasting tutorial: forecasting based on actuals

In this part of the tutorial you create a forecast based on actuals.

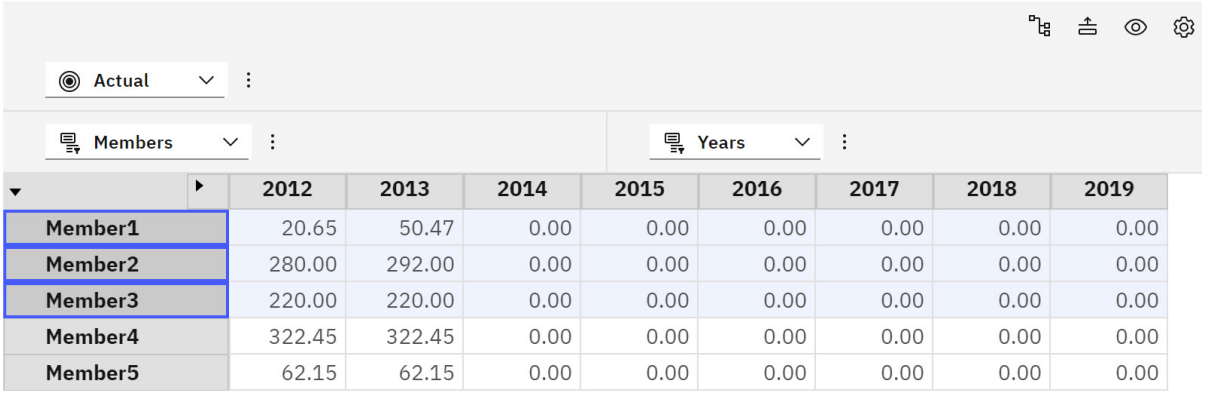
### Before you begin

This tutorial is meant to be followed in sequential order. Each part assumes that the previous part was completed.

**Note:** This part of the tutorial creates and saves forecast data. Unlike the preview you created in "Forecasting tutorial: previewing results" on page 574, when you click **Forecast** (as opposed to **Preview**), you modify the data on your Planning Analytics database.

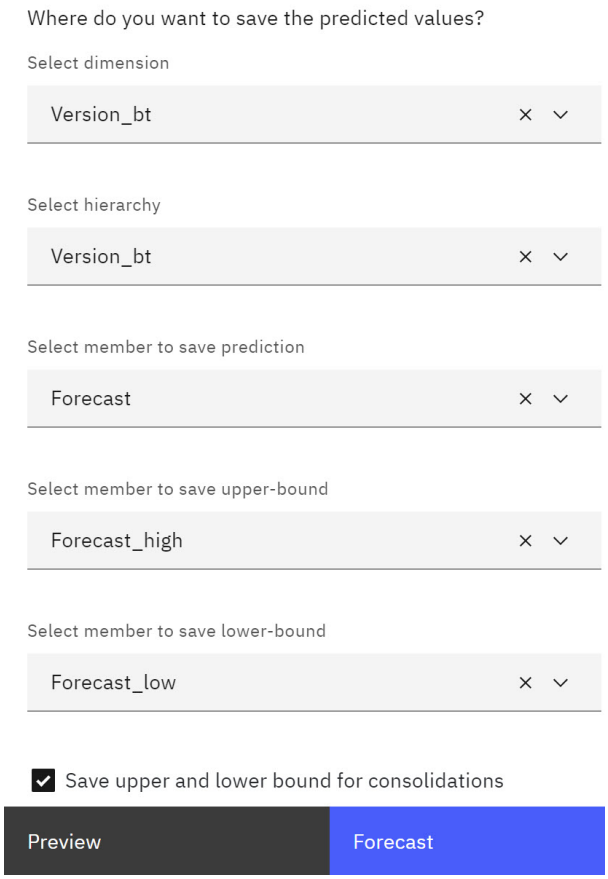
### Procedure

1. Configure the same Exploration view of the `Tutorial_<your initials>` cube as in the previous section:
   - Year on the columns
   - Members on the rows
   - Version on the context area, with the Actual member displayed
2. Select **Member1**, **Member2** and **Member3** on the row dimension.




| | 2012 | 2013 | 2014 | 2015 | 2016 | 2017 | 2018 | 2019 |
|---|---|---|---|---|---|---|---|---|
| Member1 | 20.65 | 50.47 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member2 | 280.00 | 292.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member3 | 220.00 | 220.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member4 | 322.45 | 322.45 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member5 | 62.15 | 62.15 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

3. Click the **Forecast** icon on the Exploration toolbar.
4. Set the **Forecast start period** to 2014.
5. Set the **Forecast end period** to 2019.
6. Enable the **Save statistical details as comments** option.
7. Click the **Advanced** tab on the **Forecast** panel.
8. Scroll down to the section titled **Where do you want to save the predicted values?** and configure the section as shown. Remember, the Version dimension name and the Version hierarchy name are appended with your initials.



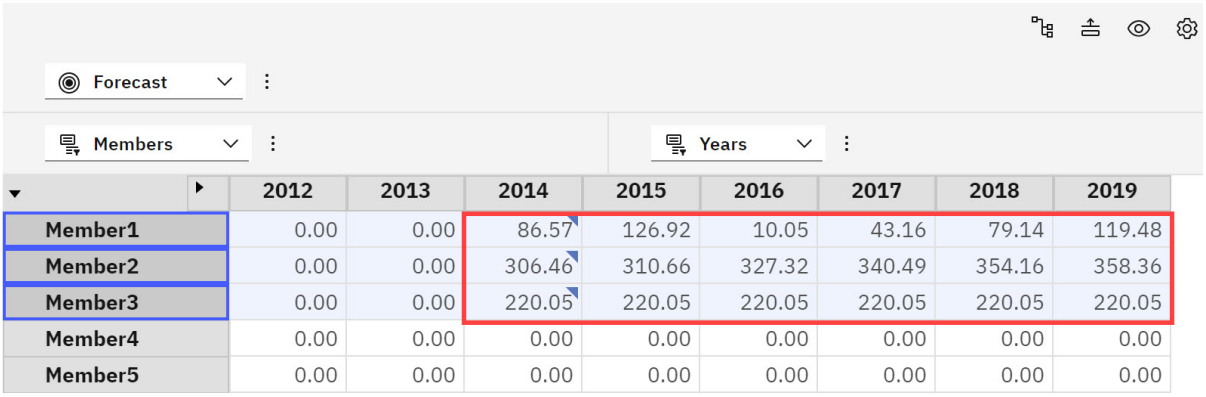

9. Click **Forecast**.

   When the forecast is done, an informational message indicates that the AI driven forecast completed successfully.

   The data in the cube refreshes automatically when the forecast is complete, but the results are not immediately visible in the view. This is because the forecast is calculated using Actual data values, but the forecast values are saved to the Forecast member.

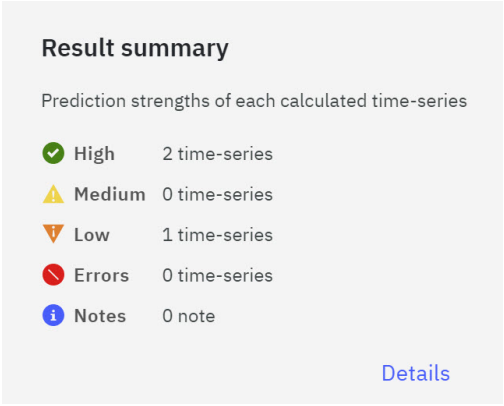
10. Click the **Version** dimension tile on the context are of the view and then select the **Forecast** member.

    You can see that forecast values were written and saved to the cube for the selected row members and the years 2014 - 2019.




| | 2012 | 2013 | 2014 | 2015 | 2016 | 2017 | 2018 | 2019 |
|---|---|---|---|---|---|---|---|---|
| Member1 | 0.00 | 0.00 | 86.57 | 126.92 | 10.05 | 43.16 | 79.14 | 119.48 |
| Member2 | 0.00 | 0.00 | 306.46 | 310.66 | 327.32 | 340.49 | 354.16 | 358.36 |
| Member3 | 0.00 | 0.00 | 220.05 | 220.05 | 220.05 | 220.05 | 220.05 | 220.05 |
| Member4 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Member5 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

The **Result summary** box and the **Details** link on the Forecast panel display information about the predictive strength of each time series in the forecast. In this case, two of the time series have a High prediction strength, while the other has a Low prediction strength.

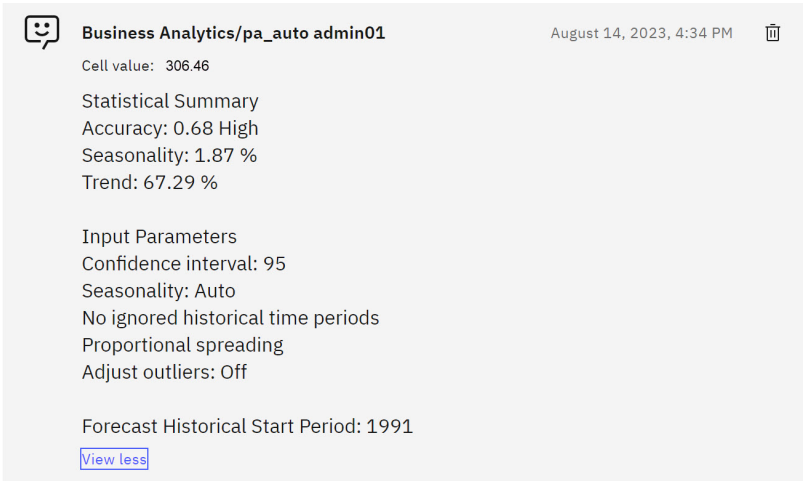



Click **Details** to see more information about the prediction strength of each time series.

Because you enabled the **Save statistical details as comments** when you configured the forecast, you can also view statistical details for the forecast directly in the Exploration view. The first cell in a forecast time series displays a blue triangle as an indicator that comments or statistical details are saved with the cell.




11. Right-click the cell in the forecast with the comment indicator, then click **Comments** to see the statistical details for the forecast on the associated row member.



12. Click **Close** to dismiss the comments.
13. Click **Save** on the Exploration toolbar.
14. Enter `Forecast_tutorial_<your initials>` as the view name.
15. Click the **Shared** tab to save the Exploration view to the Planning Analytics database Shared folder. When a view with forecast data is saved to the Shared folder or as a public view on the Planning Analytics database, other analysts can explore the forecast you create.
16. Click **Save**.

## Forecasting tutorial: view forecast data with a Line visualization

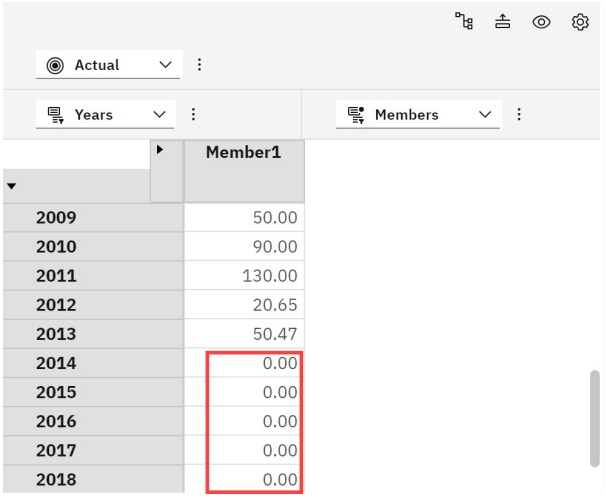
You can use a Line visualization to view and share a graphical representation of forecasting results.

### Before you begin

This tutorial is meant to be followed in sequential order. Each part assumes that the previous part was completed.

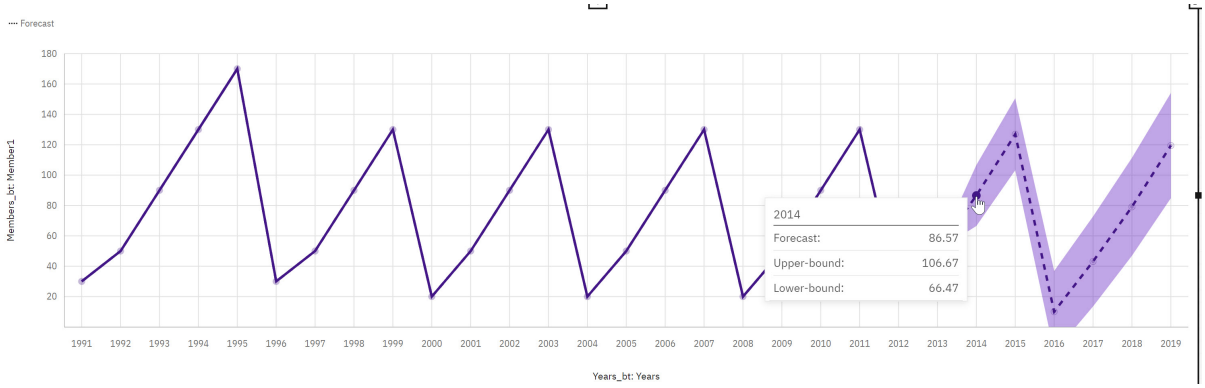
### Procedure

1. Open the `Forecast_tutorial_<your initials>` view that you created in the previous tutorial exercise.
2. Configure the view as shown:
   - All `Years` members on the rows.
   - `Member1` from the Members dimension on the columns.
   - The Version dimension on the context area, with the `Actual` member selected.




   Note that the view is displaying values for the `Actual` member, so there are no forecast values displayed. The forecast values are saved to the `Forecast` member, which is not part of this Exploration view.
3. On the toolbar, click **Exploration**, then scroll down and click the **Line** visualization.

   A Line visualization opens, combining `Actual` and `Forecast` data for `Member1`. Click any of the forecast data points to see the details and to reveal the confidence envelope for the forecast.




The Line visualization for the forecast combines `Actual` and `Forecast` data by detecting the presence of an existing forecast and setting the relevant **Forecast fields** for the visualization.

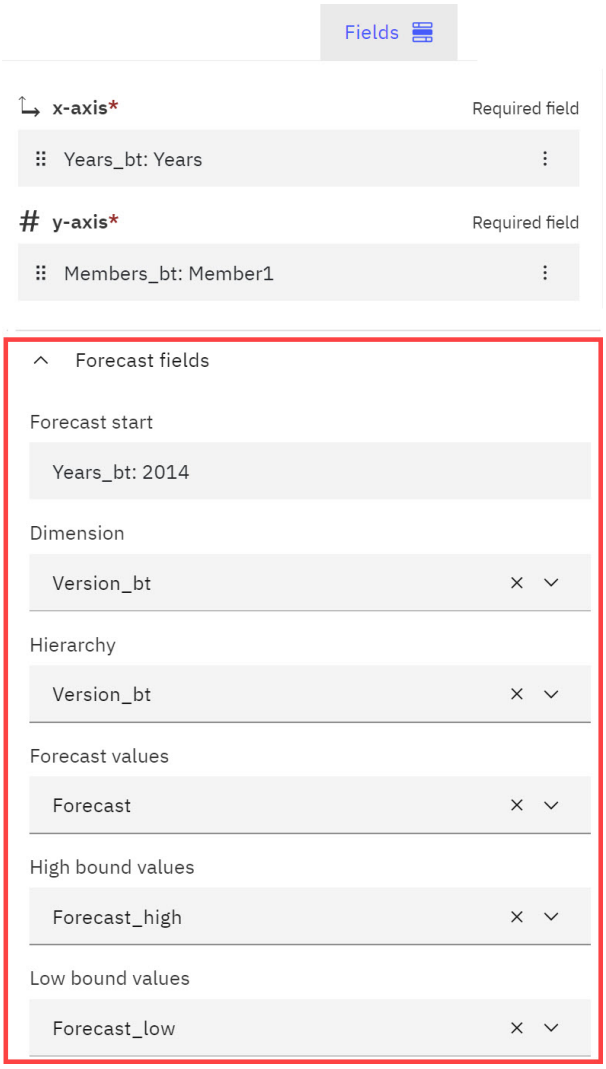
If the Line visualization does not display forecast data, but you know that forecast data does exist for the view, you can configure the visualization fields to display the forecast.

4. Click visualization, then click the **Fields** tab.
5. Expand the **Forecast fields** section.
6. Set the **Forecast start** to the dimension and member of the start of your forecast period.
7. Set **Dimension** to the dimension that contains the member that stores forecast values.
8. Set **Hierarchy** to the dimension hierarchy that contains the member that stores forecast values.
9. Set **Forecast** values to the member that stores forecast values.
10. Set **High bound** values to the member that stores high bound values for the forecast.

    Multivariate forecasts do not use a high bound. This field must be left empty for a multivariate forecast.
11. Set **Low bound** values to the member that stores low bound values for the forecast.

    Multivariate forecasts do not use a low bound. This field must be left empty for a multivariate forecast.

    Using the tutorial sample data, the fields would be configured lie this:




## Forecasting tutorial: outlier detection and adjustment

An outlier is any point in the last 20% of historical data that is outside of the confidence range for that point in history. The Watson Studio library automatically identifies outliers in your historical data. When forecasting in Planning Analytics Workspace you can choose to adjust for outliers in your forecast. The forecast model is corrected by using a "normalized" data point for the outlier to increase the forecast accuracy. The actual data point that is identified as an outlier is not modified.

### Before you begin

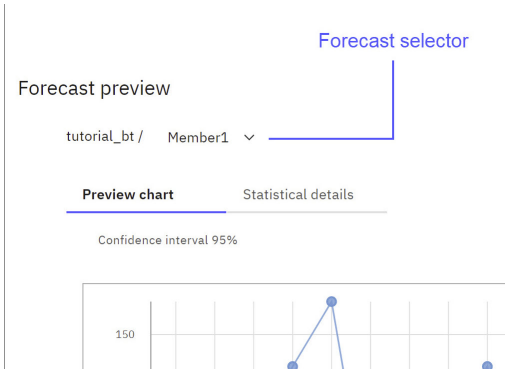
This tutorial is meant to be followed in sequential order. Each part assumes that the previous part was completed.

### About this task

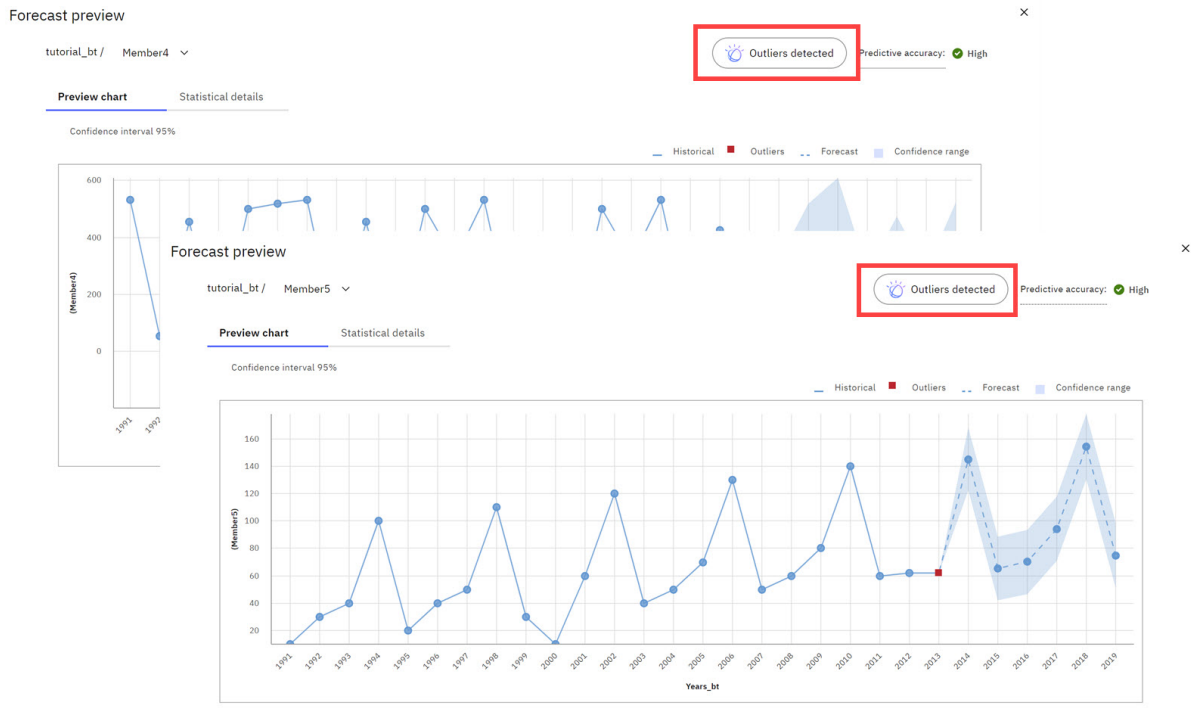
When forecasting, many models are evaluated to determine which one proves to be the most accurate overall, then the one with highest overall accuracy is selected. Although not displayed in Planning Analytics Workspace, the points along the model history have a confidence interval similar to that displayed with the forecasted values. Where many points along the model history may map nearly exactly to the actual historical points received, there may be a subset that sit outside of the confidence level, and it is these points that are identified as outliers. in a forecast

## Procedure

1. Configure the Exploration view of the `Tutorial_<your initials>` cube as in the previous section:
   - Years on the columns
   - Members on the rows
   - Version on the context area, with the Actual member displayed
2. Select all row members, then click the **Forecast** icon on the Exploration toolbar.
3. Click **Preview**.
4. Use the **Forecast selector** to cycle through the previews for all five row members.

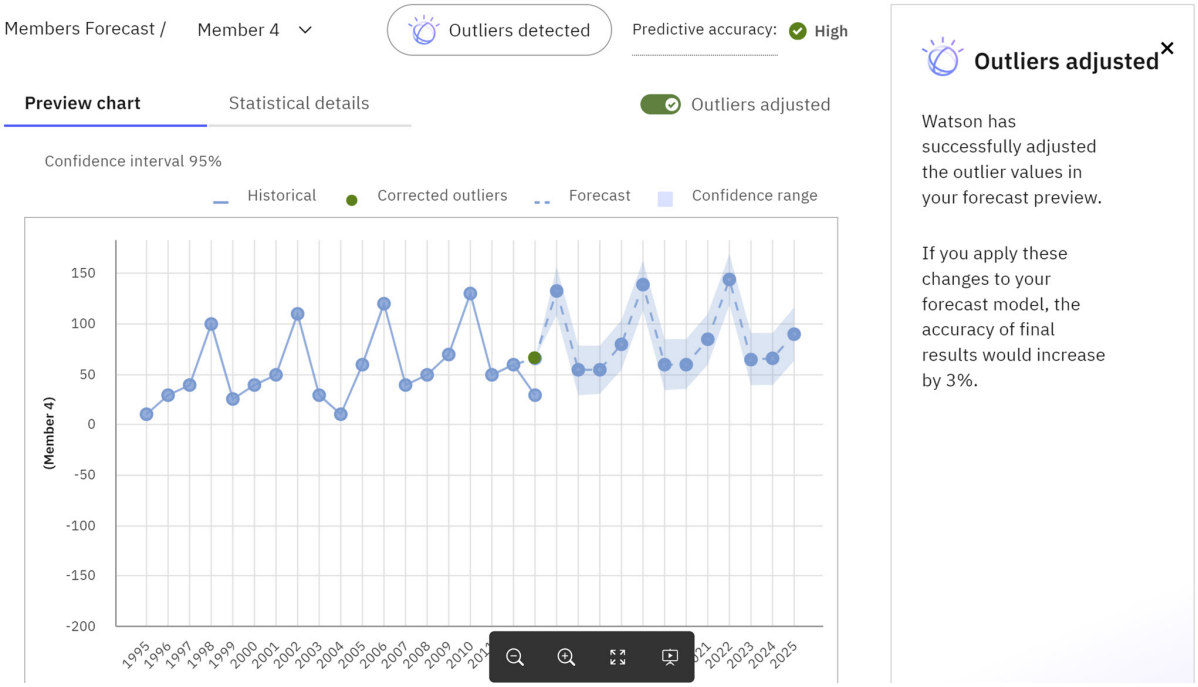



The forecast previews for Member1, Member2, and Member3 have varying degrees of predictive accuracy, but none of these previews contain outliers. Conversely, the previews for Member4 and Member 5 contain outliers, as indicated by the **Outliers detected** button on the preview.

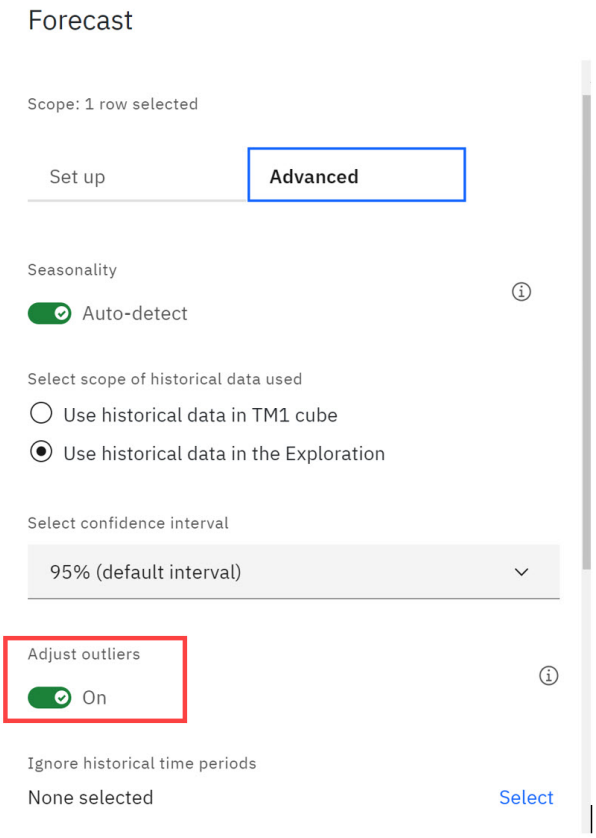



5. On the preview for Member4, click the **Outliers detected** button to open the **Outliers detected** panel.
6. Click **Adjust outliers** to adjust the outlier values in the forecast preview.

   When outliers are adjusted in this way, an **Outliers adjusted** toggle is available on the forecast preview. You can toggle this option on or off to see the impact of outlier adjustment on your forecast preview.




Alternatively, you can enable the Advanced option to automatically adjust outliers in a forecast. Click the **Advanced** tab on the **Forecast** page, then toggle the **Adjust outliers** option to the On position.




When **Adjust outliers** is enabled, outliers are automatically corrected before a preview is generated. You don't see an **Outliers adjusted** option when a preview is generated with **Adjust outliers** enabled, but the preview does show where the outliers exists in the historical data and how the preview was adjusted to compensate for the outliers.

## Forecasting tutorial: data spreading methods for forecasts

This part of the tutorial illustrates the use of different data spreading in forecasts. In particular, you'll compare the Proportional spread method to the Relative Proportional spread method and determine which is most beneficial to your business needs.

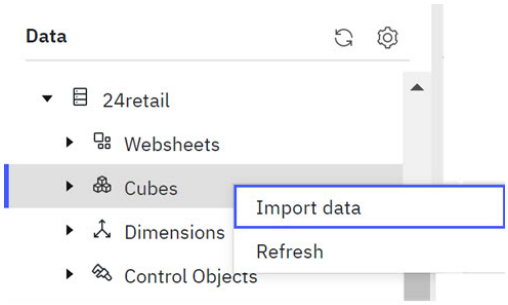
### Before you begin

This tutorial is meant to be followed in sequential order. Each part assumes that the previous part was completed.

This tutorial uses the `pa_data_spreading_demo.csv` sample data file, which you downloaded in "Forecasting tutorial: preparing data" on page 570

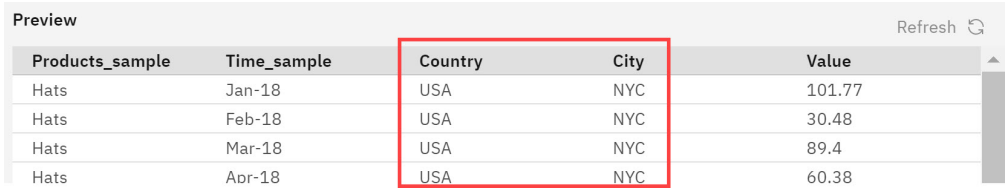
### Procedure

1. Create a new Planning Analytics Workspace book.
2. Expand the **24Retail** database on the **Data** tree.
3. Right-click **Cubes**, then click **Import data**.




4. On the **Import data** window, click **New cube**.
5. Name the cube `Spreading_<your initials>`, using your initials as a suffix.
6. Upload the `pa_data_spreading_demo.csv`file, then click **Next**.
7. Examine the preview of the `pa_data_spreading_demo.csv` file, then click **Next**.

   Note that the file contain two columns that contain geographic information. You'll set mapping options to use these two columns to create consolidations in a Region dimension.




| Products_sample | Time_sample | Country | City | Value |
|---|---|---|---|---|
| Hats | Jan-18 | USA | NYC | 101.77 |
| Hats | Feb-18 | USA | NYC | 30.48 |
| Hats | Mar-18 | USA | NYC | 89.4 |
| Hats | Apr-18 | USA | NYC | 60.38 |

8. Ensure that the **Parent-Child** mapping format option is selected on the **Map data** panel.
9. Click **Overwrite existing values**.
10. Click **Save process as**, then enter `Spreading_data_import_<your initials>` as the process name.

    Now it's time to map the columns in the `pa_data_spreading_demo.csv` file to Planning Analytics dimensions. This is done in the **Data source mapping** section of the **Map data** panel.

    a) Click **Add dimension**. The source data has columns that represent three cube dimensions, but the default mapping accommodates only two dimensions. You must add the third dimension.

    b) Click the first **Enter dimension name** cell and enter `Product_<your initials>`.

    c) Click the corresponding **Select mapping cell** on the **Child** column, then click **Products_sample**.

    d) Click the next **Enter dimension name** cell and enter `Time_<your initials>`.

    e) Click the corresponding **Select mapping cell** on the **Child** column, then click **Time_sample**.

f) Click the final **Enter dimension name** cell and enter `Region_<your initials>`.

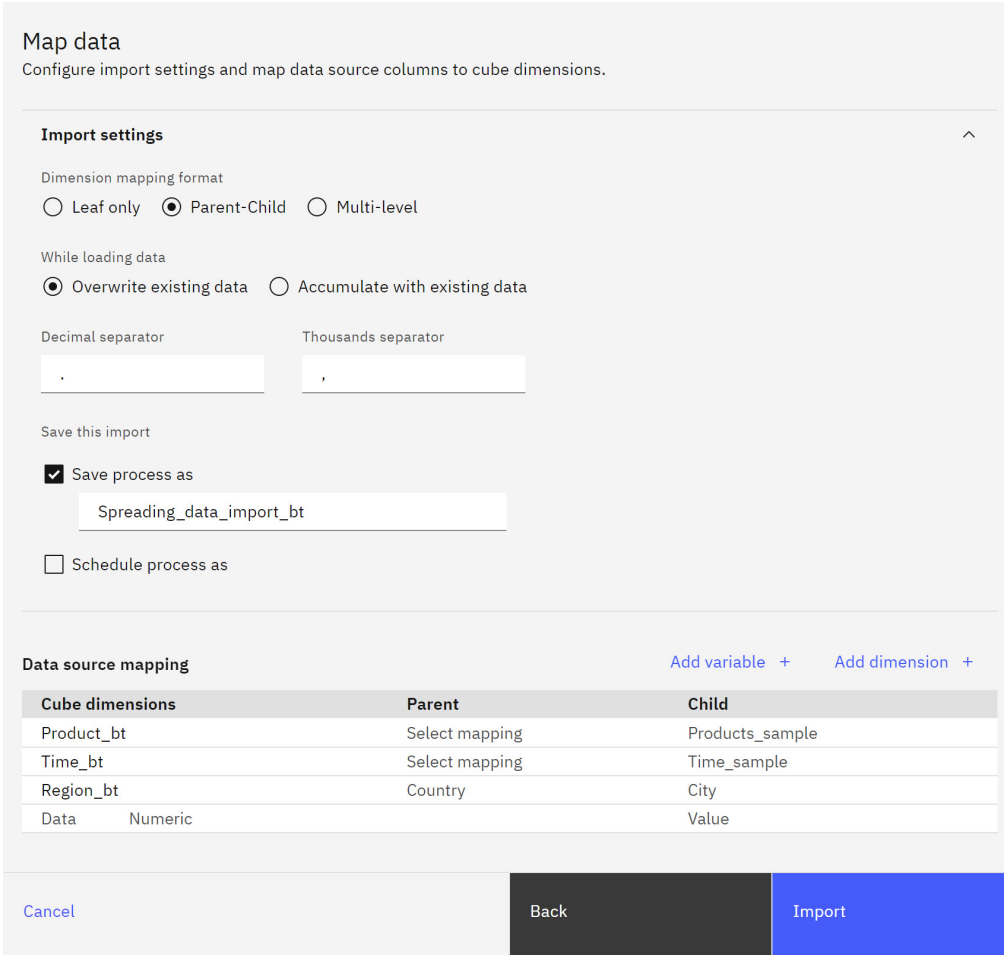
g) Click the corresponding **Select mapping cell** on the **Parent** column, then click **Country**.

h) Click the corresponding **Select mapping cell** on the **Child** column, then click **City**.

The last column in the data source is correctly identified as Data of Numeric type, but it isn't yet mapped.

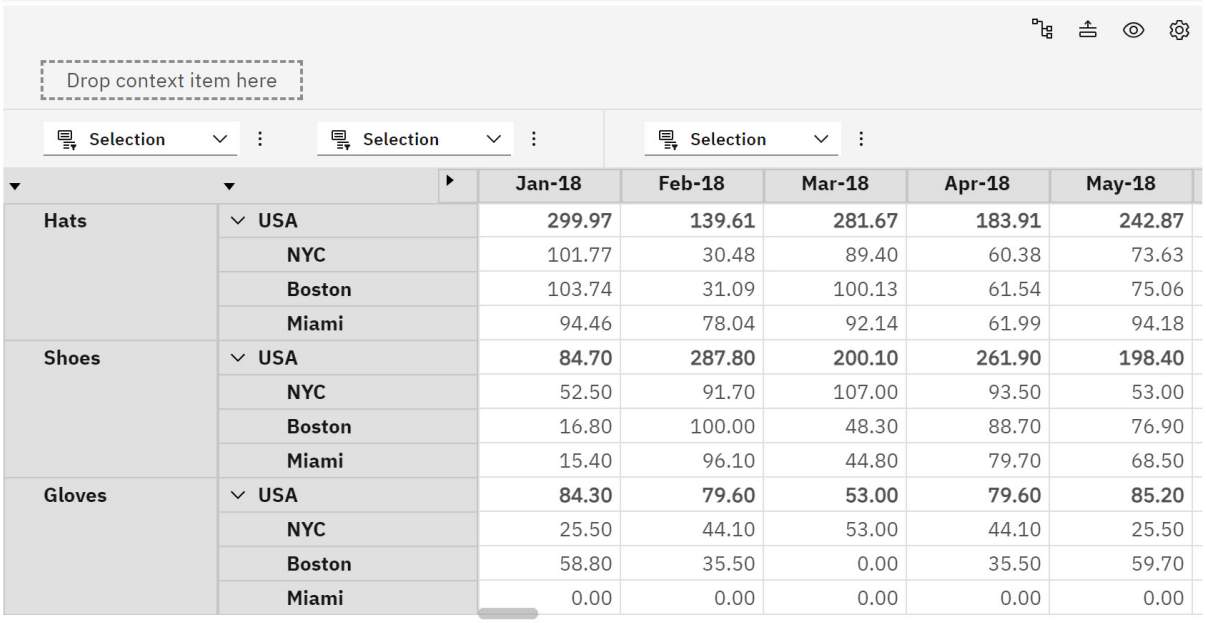
i) Click the **Select mapping cell** on the **Child** column that corresponds to Data, then click **Value**.

The **Map data** panel should now look like this:




11. Click **Import**.
12. Click **OK** after you review the import details.

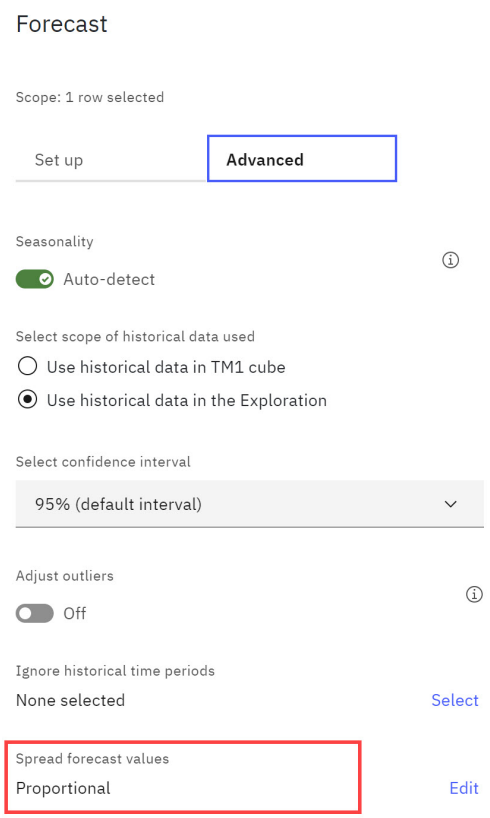
    The `Spreading_<your initials>` cube and its associated dimensions are created on the 24retail database. The default view of the cube opens in a new book.
13. Drag the dimension tiles to configure a view with `Product_<your initials>`and `Region_<your initials>` dimensions stacked on the rows and `Time_<your initials>` on the columns.




| | | Jan-18 | Feb-18 | Mar-18 | Apr-18 | May-18 |
|---|---|---|---|---|---|---|
| Hats | USA | 299.97 | 139.61 | 281.67 | 183.91 | 242.87 |
| | NYC | 101.77 | 30.48 | 89.40 | 60.38 | 73.63 |
| | Boston | 103.74 | 31.09 | 100.13 | 61.54 | 75.06 |
| | Miami | 94.46 | 78.04 | 92.14 | 61.99 | 94.18 |
| Shoes | USA | 84.70 | 287.80 | 200.10 | 261.90 | 198.40 |
| | NYC | 52.50 | 91.70 | 107.00 | 93.50 | 53.00 |
| | Boston | 16.80 | 100.00 | 48.30 | 88.70 | 76.90 |
| | Miami | 15.40 | 96.10 | 44.80 | 79.70 | 68.50 |
| Gloves | USA | 84.30 | 79.60 | 53.00 | 79.60 | 85.20 |
| | NYC | 25.50 | 44.10 | 53.00 | 44.10 | 25.50 |
| | Boston | 58.80 | 35.50 | 0.00 | 35.50 | 59.70 |
| | Miami | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

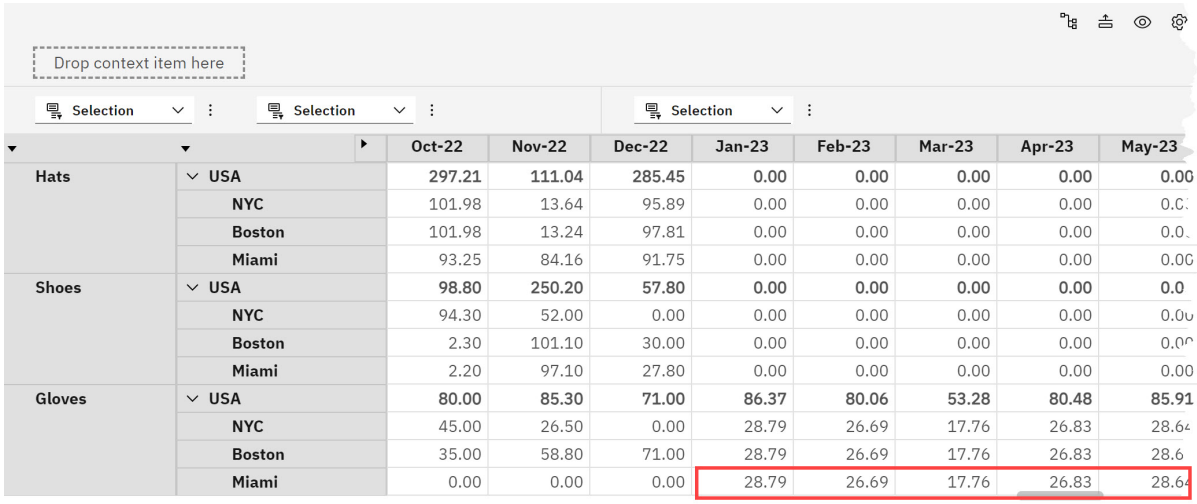
Review the data in the view and note that there are no glove sales in Miami for any time period.

14. Select the **Gloves, USA** row, then click the **Forecast** icon on the Exploration toolbar.
15. On the **Forecast** panel, set the **Forecast period start** to Jan-23. The **Forecast period end** should automatically be identified as Dec-23.
16. Click **Preview** to preview the forecast. In this exercise, do not correct any outliers.
17. Dismiss the preview, then click the **Advanced** tab on the Forecast panel. Notice that by default, the forecast operation uses the **Proportional** data spreading method to forecast values.




18. Click **Forecast**.

Examine the forecast values and note that the Proportional data spreading method creates forecast values for gloves in Miami, while there were no glove sales in that location prior to the forecast period.



Drop context item here

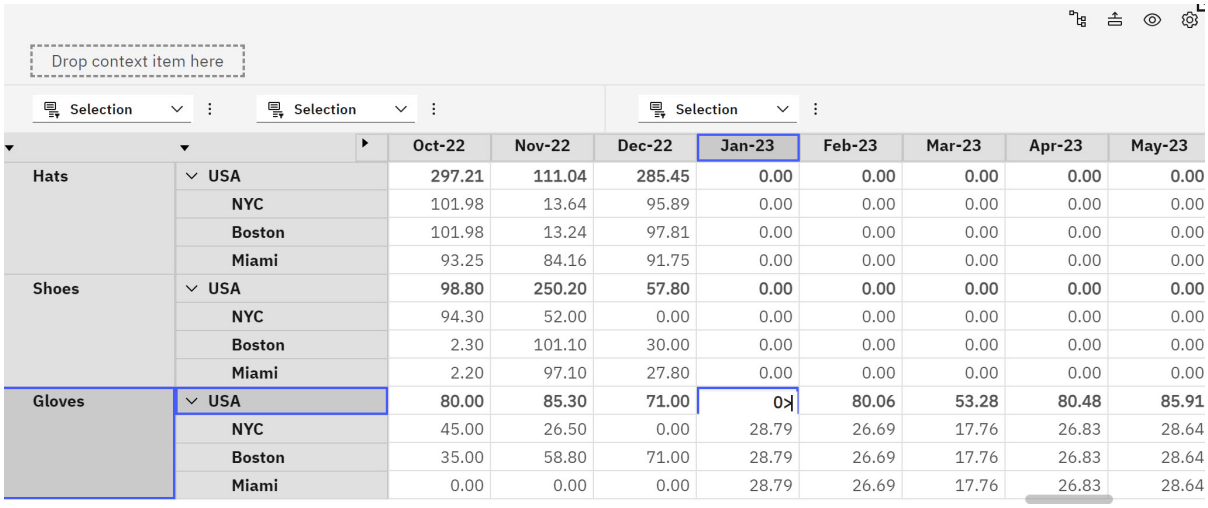
Selection | Selection | Selection

| | | Oct-22 | Nov-22 | Dec-22 | Jan-23 | Feb-23 | Mar-23 | Apr-23 | May-23 |
|---|---|---|---|---|---|---|---|---|---|
| Hats | USA | 297.21 | 111.04 | 285.45 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | NYC | 101.98 | 13.64 | 95.89 | 0.00 | 0.00 | 0.00 | 0.00 | 0.0 |
| | Boston | 101.98 | 13.24 | 97.81 | 0.00 | 0.00 | 0.00 | 0.00 | 0.0 |
| | Miami | 93.25 | 84.16 | 91.75 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Shoes | USA | 98.80 | 250.20 | 57.80 | 0.00 | 0.00 | 0.00 | 0.00 | 0.0 |
| | NYC | 94.30 | 52.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.0 |
| | Boston | 2.30 | 101.10 | 30.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.0 |
| | Miami | 2.20 | 97.10 | 27.80 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Gloves | USA | 80.00 | 85.30 | 71.00 | 86.37 | 80.06 | 53.28 | 80.48 | 85.91 |
| | NYC | 45.00 | 26.50 | 0.00 | 28.79 | 26.69 | 17.76 | 26.83 | 28.64 |
| | Boston | 35.00 | 58.80 | 71.00 | 28.79 | 26.69 | 17.76 | 26.83 | 28.6 |
| | Miami | 0.00 | 0.00 | 0.00 | 28.79 | 26.69 | 17.76 | 26.83 | 28.64 |

This is because consolidations in the time series (in this case, USA) are forecast using proportional spreading, but values for members of the consolidations are created by equally spreading the consolidation value to all members. In this example the result is unsatisfactory because historically there have been no glove sales in Miami; the forecast should not predict future glove sale.

Fortunately, you can correct this forecast by changing the forecast data spreading method from Proportional to Relative Proportional.

19. Click the cell at the intersection of **Gloves, USA** and **Jan-23**, then enter **0>** to clear the results of the forecast.



Drop context item here

Selection | Selection | Selection

| | | Oct-22 | Nov-22 | Dec-22 | Jan-23 | Feb-23 | Mar-23 | Apr-23 | May-23 |
|---|---|---|---|---|---|---|---|---|---|
| Hats | USA | 297.21 | 111.04 | 285.45 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | NYC | 101.98 | 13.64 | 95.89 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | Boston | 101.98 | 13.24 | 97.81 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | Miami | 93.25 | 84.16 | 91.75 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Shoes | USA | 98.80 | 250.20 | 57.80 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | NYC | 94.30 | 52.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | Boston | 2.30 | 101.10 | 30.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | Miami | 2.20 | 97.10 | 27.80 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Gloves | USA | 80.00 | 85.30 | 71.00 | 0> | 80.06 | 53.28 | 80.48 | 85.91 |
| | NYC | 45.00 | 26.50 | 0.00 | 28.79 | 26.69 | 17.76 | 26.83 | 28.64 |
| | Boston | 35.00 | 58.80 | 71.00 | 28.79 | 26.69 | 17.76 | 26.83 | 28.64 |
| | Miami | 0.00 | 0.00 | 0.00 | 28.79 | 26.69 | 17.76 | 26.83 | 28.64 |

20. If the Forecast panel is not visible, click .
21. Click the **Advanced** tab on the Forecast panel.
22. Click **Edit** on the Spread forecast values section of the Advanced tab.



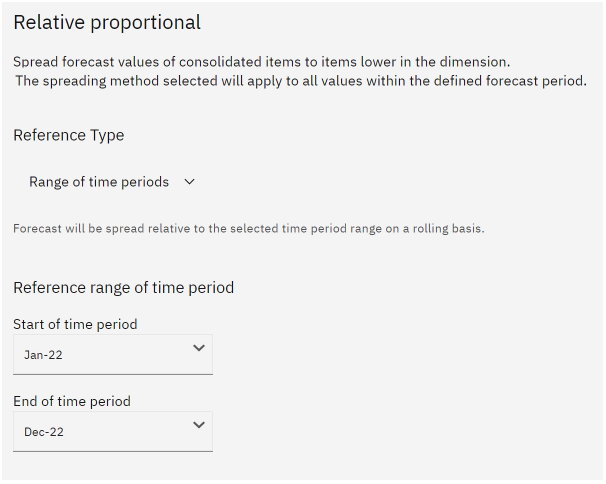
Spread forecast values

Proportional Edit

23. Click **Relative proportional** on the Spread data options dialog.
24. Select **Range of time periods** from the **Reference Type** list.

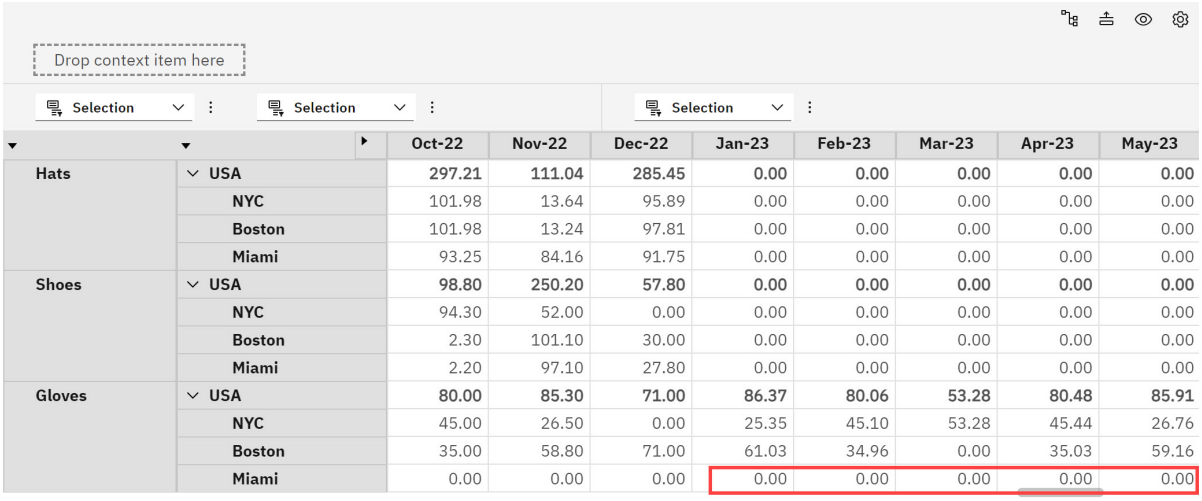
You can define the time period you want to use as the reference values for the proportional data spread. In this case, you want to spread forecast values relative to the values for the same region and product in the previous year.

25. From the **Start of time period** list, select **Jan-22**.
26. From the **End of time period** list, select **Dec-22**.




27. Click **Apply**.
28. Click **Forecast**.

    Because the forecast now spreads values proportional to existing values in a defined range, there are no forecast values for gloves in Miami, as there were no values for that region and product in the reference range.




| | | Oct-22 | Nov-22 | Dec-22 | Jan-23 | Feb-23 | Mar-23 | Apr-23 | May-23 |
|---|---|---|---|---|---|---|---|---|---|
| Hats | USA | 297.21 | 111.04 | 285.45 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | NYC | 101.98 | 13.64 | 95.89 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | Boston | 101.98 | 13.24 | 97.81 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | Miami | 93.25 | 84.16 | 91.75 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Shoes | USA | 98.80 | 250.20 | 57.80 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | NYC | 94.30 | 52.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | Boston | 2.30 | 101.10 | 30.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| | Miami | 2.20 | 97.10 | 27.80 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Gloves | USA | 80.00 | 85.30 | 71.00 | 86.37 | 80.06 | 53.28 | 80.48 | 85.91 |
| | NYC | 45.00 | 26.50 | 0.00 | 25.35 | 45.10 | 53.28 | 45.44 | 26.76 |
| | Boston | 35.00 | 58.80 | 71.00 | 61.03 | 34.96 | 0.00 | 35.03 | 59.16 |
| | Miami | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

# Use Decision Optimization (Cloud only)

You can use Decision Optimization (IBM ILOG CPLEX) with IBM Planning Analytics Workspace on Cloud to deliver prescriptive analytics capabilities to help you make better decisions and deliver improved business outcomes.

With the Cloud Pak for Data Platform integration configured, and the prerequisite services IDs and views in place, analysts can view and run optimization jobs within the configured deployment space directly from Planning Analytics Workspace. Additionally, they can configure action buttons to run any of the jobs of a deployment space, allowing any Planning Analytics Workspace on Cloud user to run optimization jobs.

Decision Optimization is not supported on Planning Analytics Workspace Local.

See Decision Optimization for full details on Decision Optimization capabilities and benefits.

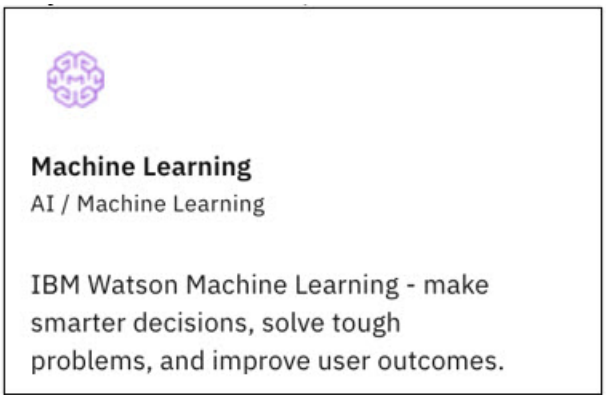
# Decision Optimization prerequisites

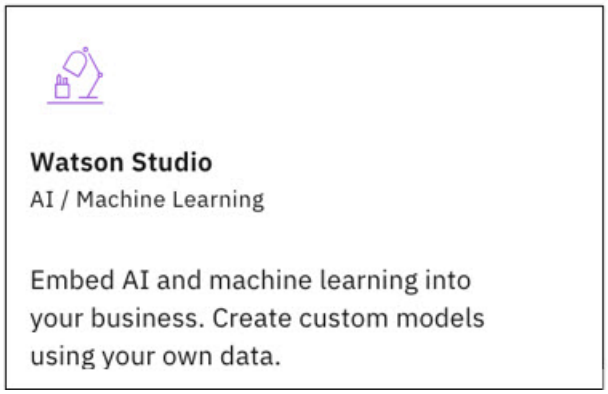
The following services, IDs, and views that must be in place before you can use Planning Analytics Workspace to view and execute Decision Optimization jobs.

## Services

Ensure that you have procured both the Machine Learning and Watson Studio service instances with IBM Cloud Pak for Data as a Service.

These are the services as they appear in Cloud Pak for Data as a Service.




## Non-interactive TM1 account

Ensure that the Non-interactive TM1 account has been created and has the appropriate access to all cubes required of the Decision Optimization you want to create. The non-interactive TM1 account is available within the Welcome Kit, which is available for download from Box@IBM. For full details on the Welcome Kit, including the non-interactive TM1 account, see the Welcome Kit on IBM Docs.

## IBM Cloud Pak for Data functional ID

There is limited group support within Cloud Pak for Data. As a result, to prevent the need for creating an ID to match every user within Planning Analytics Workspace, Decision Optimization requires a single functional ID to be created within Cloud Pak for Data, which will be used by all users to run jobs.

## Views

Decision Optimization leverages views (both MDX and native views) to read and write data to and from the TM1 database. The Planning Analytics modeler or analyst should communicate with the author of the optimization logic to understand the data they require and create the appropriate views. This will allow them to create the required Data Assets to interact with the TM1 data.

# Create a Decision Optimization job

The general procedure described here illustrates the integration between the technologies that enable Decision Optimization in IBM Planning Analytics.

## About this task

To execute your own optimization within IBM Cloud Pak for Data as a Service, you must create a project, a connection, data assets, the Decision Optimization experiment, and a model within the project. You can then deploy these assets (model, data assets and connection) to the deployment space. The model can then be deployed and a job created.

Each of the steps listed here includes a link to the comprehensive documentation on IBM Docs that fully explains and illustrates the required action.

## Procedure

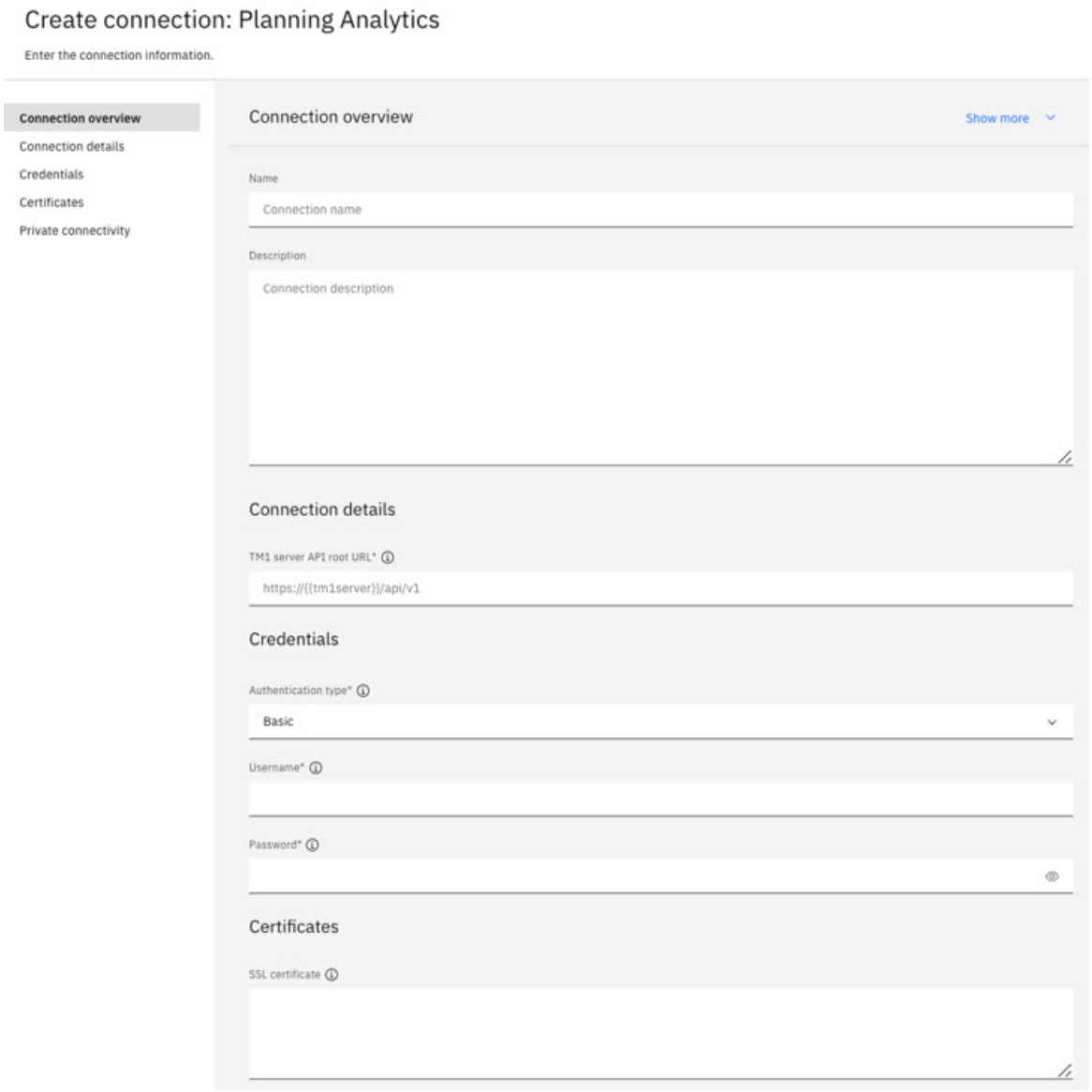
1. Create a project.

   If you don't already have one, create a new project within Cloud Pak for Data as a Service. Open the Projects page, then click **New project +**. See Creating a project for full detail on how to create in Cloud Pak for Data as a Service.

2. Create a connection.

   a) Within the project, click **Add to project +**, then click **Connection**.

   b) Click **Planning Analytics** in the list of available connections.




   c) Enter the following connection information on the **Connection overview** page.

   **TM1 server API root URL**
   The TM1 Database endpoint is created by concatenating `https://` *`+hostname+`*`/tm1/api/`*`+tm1_database_name+`*`/api/v1`. For example: `https://planning-analytics.ibmcloud.com/tm1/api/PlanningSample/api/v1/` connects to Planning Analytics database named PlanningSample (or Planning Sample – spaces can be trimmed out of the url).

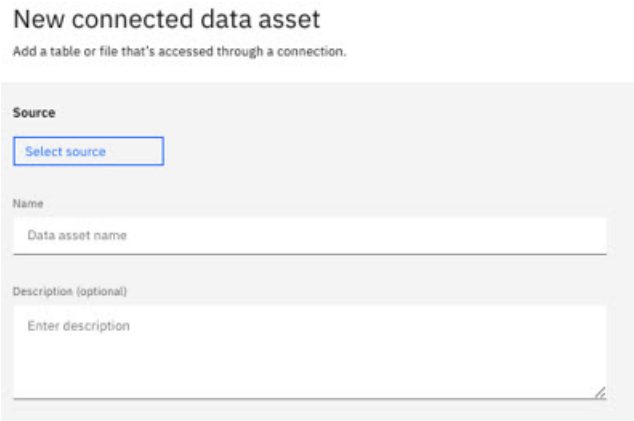
   **Authentication Type**
   Choose **Basic** or **CAM**. CAM provides secured communications with your database.

**Username**
This is the non-interactive ID identified in the Welcome Kit. Ensure this user has permissions for the database you are connecting to.

**Password**
This is the password of the non-interactive ID.

See Adding connections to projects for full details on adding a connection.

3. Create a data asset.
   a) Within the project, click **Add to project +**, then click **Connected data** when you are prompted to choose an asset type.
   b) Select the connection you created in Step 2.
   c) On the **New connected data asset** screen, click **Select source**.




   d) Navigate to the cube, then the view that you wish to expose as this data asset. This view will later be used as an input and/or output for your decision optimization job.

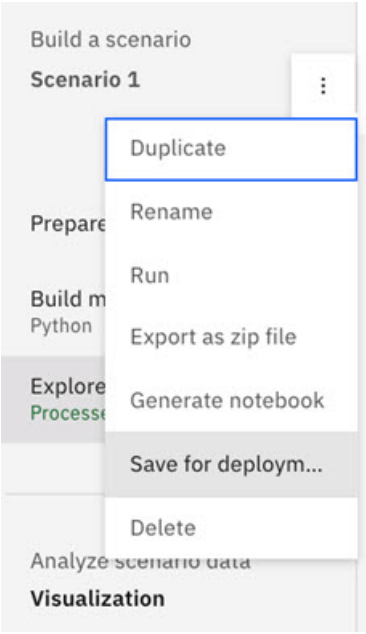
      For full details on creating a data asset, see Adding data from a connection to a project.

4. Create a Decision Optimization experiment.
   a) Within the project, click **Add to project +**, then click **Decision Optimization**.
   b) Create the experiment and create a new deployment space if one doesn't exist yet. You can then import the data assets you set up earlier as inputs in the **Prepare data** stage,
   c) Create the model (for example, author Python source code).

      For details on how to author an optimization model, see Decision Optimization experiment views and scenarios.

      A tutorial that walks you through the creation of a model is available at Solving and analyzing a model.

5. Save the model for deployment.
   a) After you successfully run the model, you can save the model for deployment by clicking the **Save for deployment** option next to the scenario.

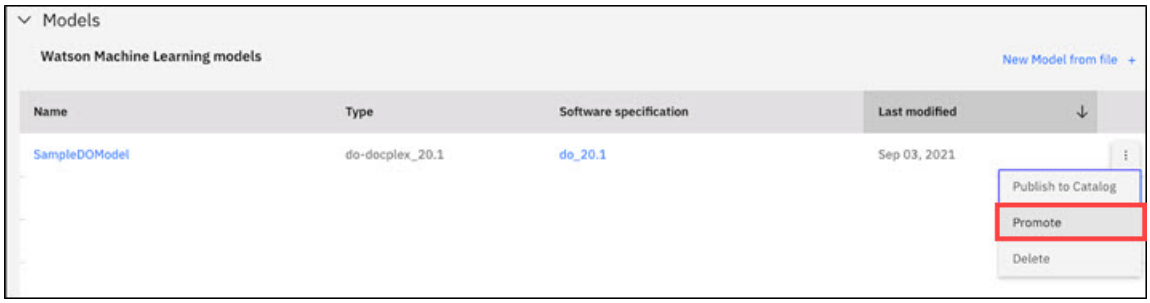



6. Promote the model to the deployment space.

   With a model now created as an asset in your project space, you can promote it and all associated assets (data assets and connections) to the deployment space you created earlier.

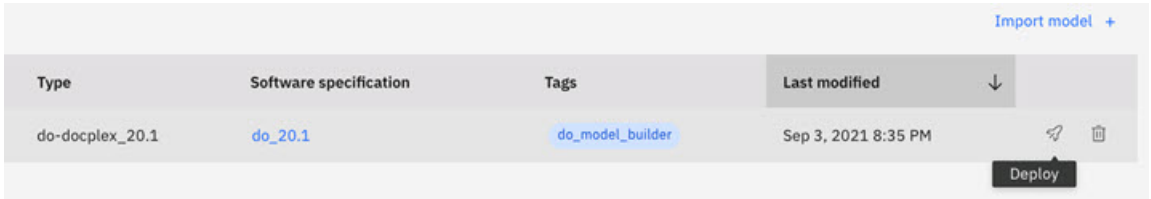
   a) Click the **Option** button ⋮ for the model

   b) Click **Promote**.




7. Deploy the model.

   a) Navigate to the deployment space using the link provided in the message that appears after promotion. Alternatively, you can select the **Deploy** action for the newly promoted model.




   b) Select the appropriate specification for this new deployment. For complete details on how to deploy assets, see Deploying assets.

8. Create a job.

   a) From the newly deployed model in your deployment space, click the **Create job +** button.

   b) Complete the wizard that appears, entering all appropriate information.

   Of particular note is the **Choose data** stage of the wizard, where you can now link your input and output views. Previously, the data for these views were effectively offline in the project space; creation of this job operationalizes the connection such that every execution is a live connection to Planning Analytics for the associated data assets.

See Running jobs for further details on creating and running jobs.

## View and run a Decision Optimization job

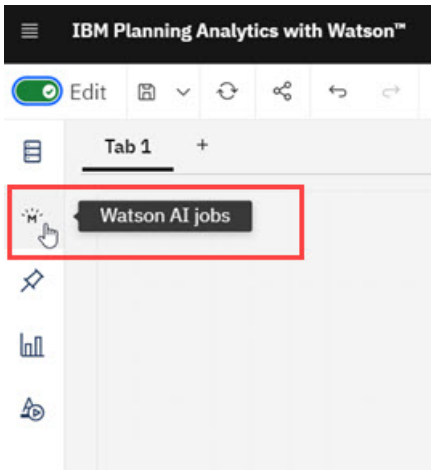
A job is typically kicked off automatically when it is first created. You can also execute a job at any time from Planning Analytics Workspace.

### About this task

You must be an administrator, modeler, or analyst to view or run a decision optimization job.

### Procedure

1. Open a book in edit mode.
2. Click the **Watson AI jobs** tab.




   The **Watson AI jobs** tab displays all the jobs available in your deployment space.

3. To view the configuration of a job, click the **Options** button ⋮ and then click **View Configuration**.
4. To run a job, click the **Options** button ⋮ and then click **Run**. You receive progress and execution messages to inform you of the status of the job execution.

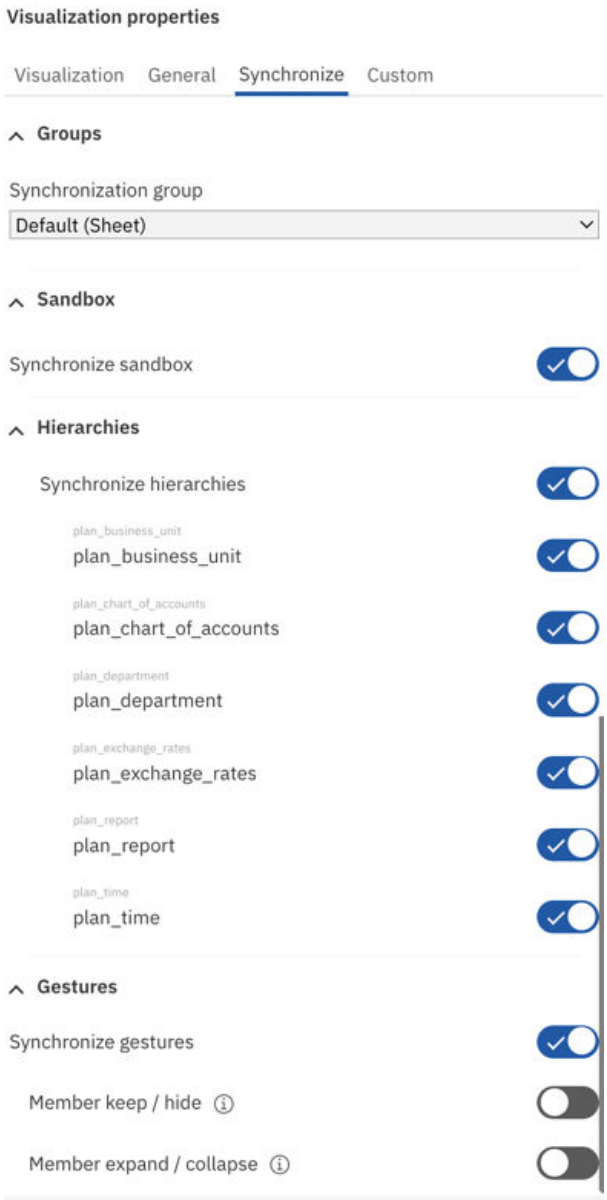
   You can also configure a button to run a Decision Optimization job. For details, see “Action Buttons ” on page 652. You do not need to be an administrator, modeler, or analyst to run a Decision Optimization job from a button on a book; any user can click the button to run the job.

# Synchronize objects in a book or sheet

Synchronization allows you to keep identical data values across objects or widgets. A change in one value impacts all of the synchronized widgets. You can set the level of synchronization that you want between views, visualizations, and other widgets at a group level, sheet or tab level, or book level.

You can set synchronization for hierarchies, sandboxes, and certain gestures under visualization properties.

**Note:** Synchronization is off by default, except for hierarchy selectors, which are synchronized.




Suppose that you have an exploration (table) view and a column chart visualization of the same view side-by-side on a sheet. You want to focus on car sales in the month of May. With synchronization enabled, when you select May in the **Month** dimension in the exploration, this selection is reflected in the column chart.

You can disable synchronization for specific hierarchies in a widget. For example, suppose that you have a **Regions** dimension with Europe selected as a hierarchy. If you want to see Europe in the exploration, but view France in the column chart, you would turn off synchronization for the **Regions** hierarchy.

Same hierarchies that are in different Planning Analytics databases can also be synchronized.

Dimension selections, such as drilling down on a member, set editing or selecting, can be synchronized. But actions such as swapping dimensions from row to column cannot be synchronized.

Similarly, if you are using sandboxes to experiment with multiple scenarios, and you want views and websheets to synchronize to a new sandbox selection, you can enable sandbox synchronization.

**Important:** Sandbox hierarchies do not synchronize across different databases.

You can also synchronize the following gestures under **Properties > Synchronize > Gestures**.

- Expand/Collapse

- Keep
- Hide (and Unhide all)

Much like synchronizing hierarchies, when you enable synchronization for a gesture, that gesture takes place in all the widgets that are a part of the same group and each widget updates its hierarchy accordingly.

**Member keep / hide**

Use **Member keep / hide** to keep or hide the hierarchy in multiple widgets. A hierarchy might be on a row in one widget and a column in another. When you synchronize **Keep** or **Hide**, the widgets update based on where the synchronized hierarchy is located in the widget.

**Member expand / collapse**

Use **Member expand / collapse** to expand or collapse a consolidated hierarchy in multiple widgets. A hierarchy might be on a row in one widget and a column in another. When you synchronize **Expand/ Collapse**, the widgets update based on where the synchronized hierarchy is located in the widget.

You can synchronize only those gestures that are symmetric. For example, if you used **Target selection** (or **F7**) to make asymmetric selections on rows and columns, you won't be able to synchronize the widgets using **Member keep / hide**. Similarly, if you expanded or collapsed rows and columns asymmetrically, you can't use **Member expand / collapse** to synchronize the widgets.

**Note:** Synchronization gestures are not available for synchronized tiles and websheets. You can synchronize gestures in selector widgets if the widgets are showing a list of members.

For information about how to enable synchronization, refer to Setting synchronization for objects.

## Group synchronization

You can assign views, visualizations, and other widgets to a synchronization group. Widgets in the same group will be synchronized together.

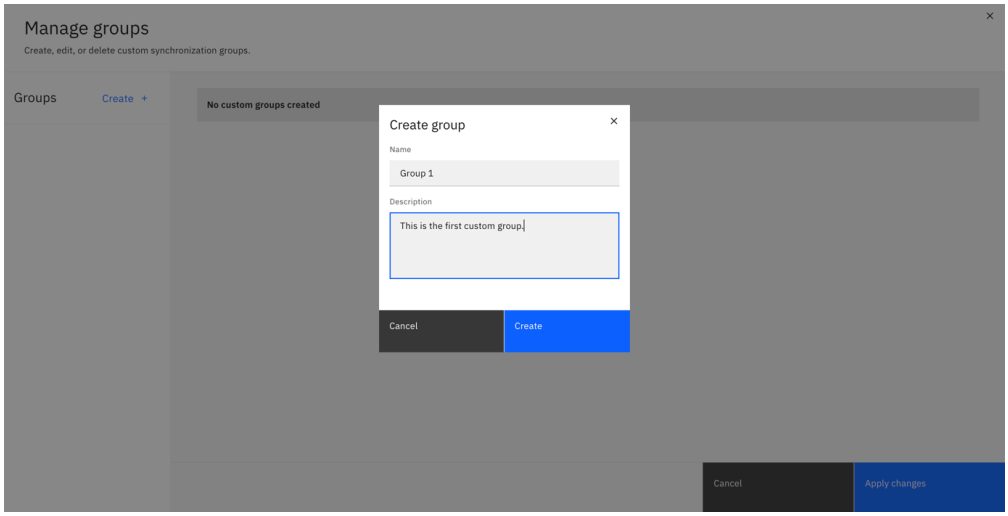
Only changes to the currently displayed members of a hierarchy in a widget are shared with other widgets. As members of a hierarchy that are displayed in one widget change, the same members display in synchronized widgets that use the same hierarchy.

Synchronization groups can span to the entire book.

### Creating a custom synchronization group

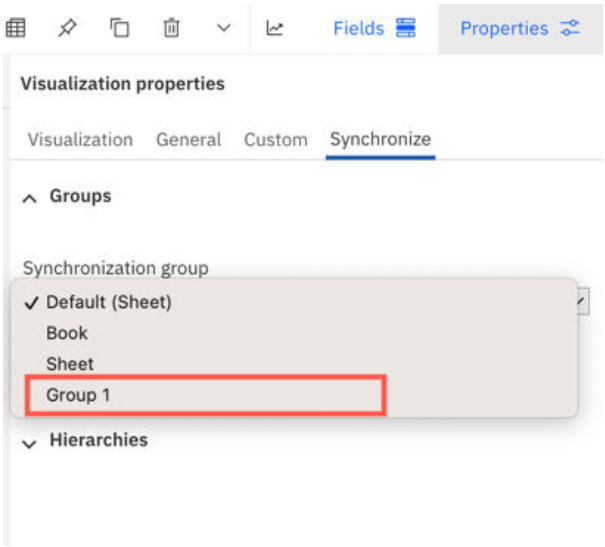
Start by creating a custom group in book properties:

1. In edit mode, click **Properties**.
2. Under **Synchronize**, click **Create and manage custom groups**.
3. Click **Create** to create a group.
4. Enter a name and description for the group, and click **Create**.

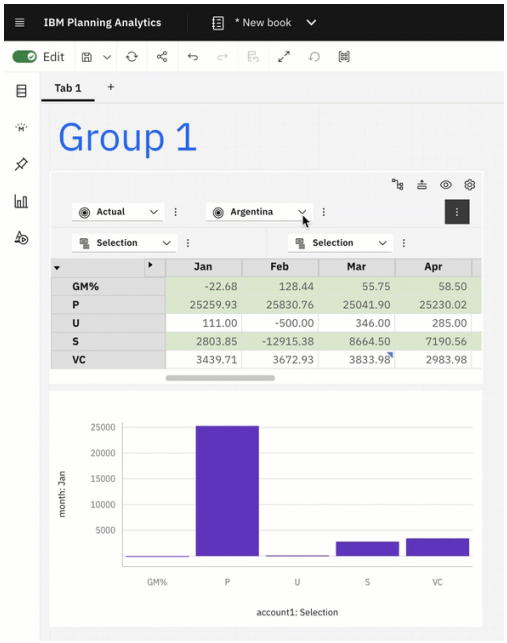



5. Click **Apply changes** to finish creating the group.

You can now add widgets to your group. Select a widget in your book and under **Properties > Synchronize> Groups > Synchronization group**, select the group you just created. Repeat this process for all the widgets that you want to assign to the group.

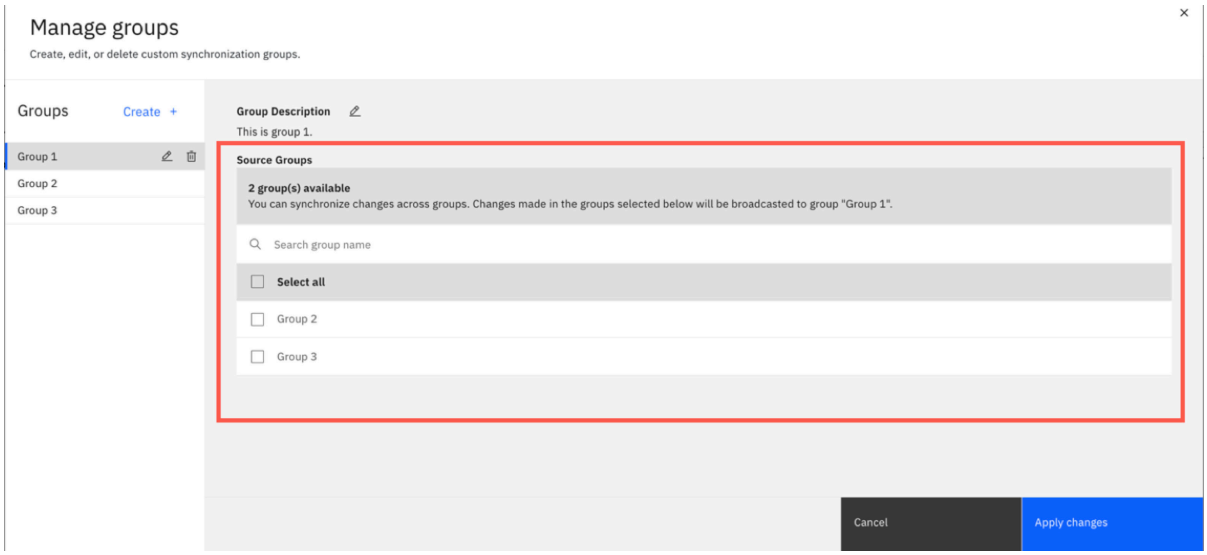



Widgets in the group are now synchronized together.

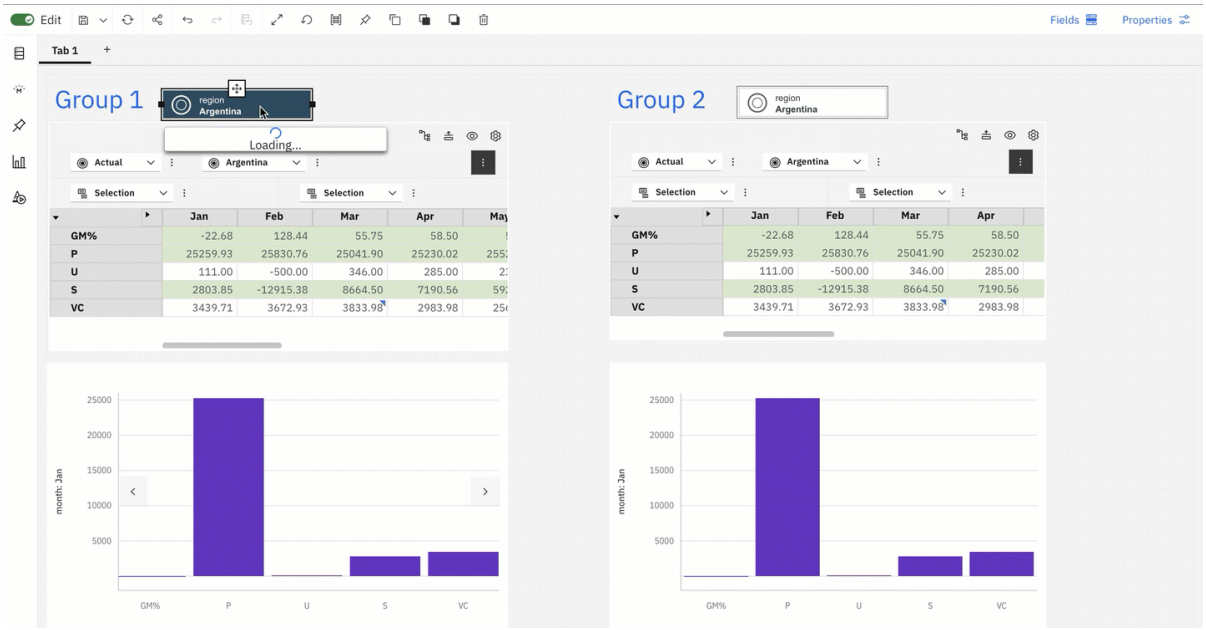



## Cascading effect in synchronization

When you have two or more synchronization groups, you can have one group impact another group to achieve an cascading effect. To do this, you can assign one or more source groups to a target group in the **Manage groups** dialog.



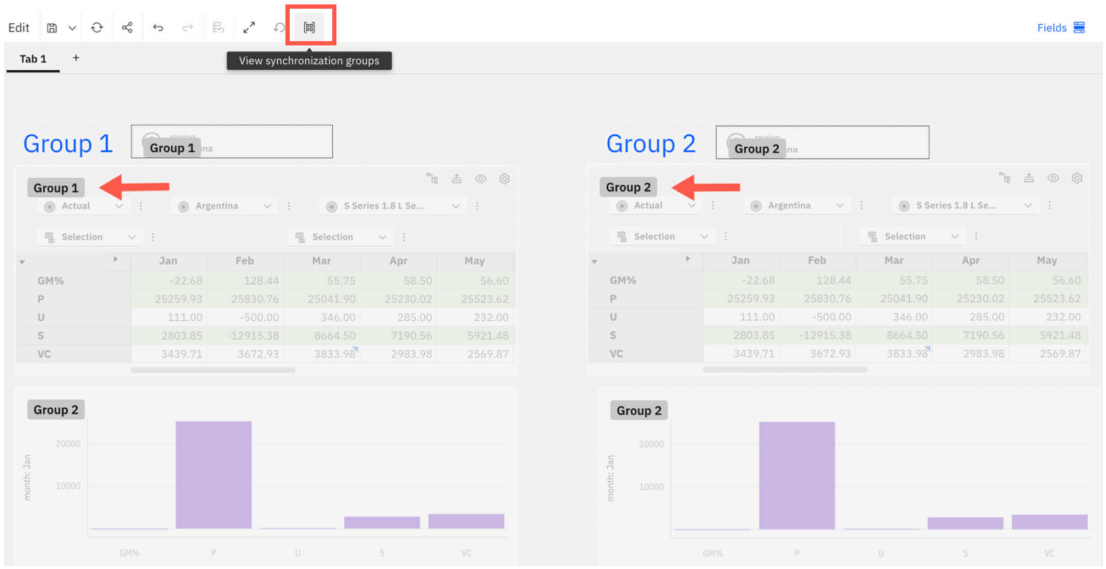

In the following example, the hierarchy change in Group 1 cascades to all the widgets in Group 1 and Group 2.




**Note:** Synchronization changes cascade only once. For example, if Group 1 is a source group for Group 2 and Group 2 is a source group for Group 3, changes in Group 1 impact only Group 2.

## View synchronization groups

Click the **View synchronization groups** icon in the toolbar to view an overlay for all widgets that are part of synchronization. The overlay displays the name of the group the widgets belong to. Click the icon again to interact with the widget.




## Action button parameters

Synchronization parameters are available for action buttons when navigating to another book or running processes that use synchronization.

### Navigation

When navigating to another book using an action button, you can choose to have synchronization settings from the source book copy over to the target book. Select the **Pass context** option under **Navigation** properties to synchronize settings.

When you pass context from the source book to the target book, default synchronization settings pass to provide context to the same synchronization group in the target book. Note, this table lists all of the possible synchronization mappings for the `default` mode and only one of them is used when navigating to the target book.

| Source book (default group) | Target book (default group) | Result |
|---|---|---|
| Book | Book | Synchronization settings are copied from the Book synchronization group in the source book to the Book synchronization group in target book. |
| Book | Sheet | Synchronization settings are copied from the Book synchronization group in the source book to all the tabs in target book. |
| Sheet | Book | Synchronization settings are copied from the current tab in the source book to the Book synchronization group in target book. |
| Sheet | Sheet | Synchronization settings are copied from the current tab in the source book to the opened tab in the target book. |
| Custom group | Book | Synchronization settings are copied from the Custom group in the source book to the Book synchronization group in the target book. |
| Custom group | Sheet | Synchronization settings are copied from the Custom group in the source book to all the tabs in the target book. |
| Book | Custom group | Synchronization settings are copied from the Book synchronization group in the source book to the Custom group in the target book. |
| Sheet | Custom group | Synchronization settings are copied from the current tab in the source book to the Custom group in the target book. |
| Custom group 1 | Custom group 2 | Synchronization settings are copied from a Custom group in the source book to another Custom group in the target book. |

**Important:** Copying settings from the default synchronization group has a higher priority than copying settings from custom groups.

Synchronization settings also copy over from the source book to the target book for any synchronization groups that are present in both books. These groups must have names that are an exact match.

**Advanced context**

When you choose to pass context with an action button, you can override the default synchronization settings described in the section above with **Advanced context**. The **Advanced context** feature lets you map a source synchronization group to a target synchronization group and select which hierarchies that are common to both groups, receive context. You can configure multiple action buttons to pass context to a single target book and drill through data easily.

Before you can map a synchronization group from the source book to the synchronization group in the target book, you need to first set up synchronization and the widgets in both books need to be of the same synchronization group. **Advanced context** supports all three synchronization groups (Book, Sheet, and Custom).

To map synchronization groups and configure context passing, follow these steps:

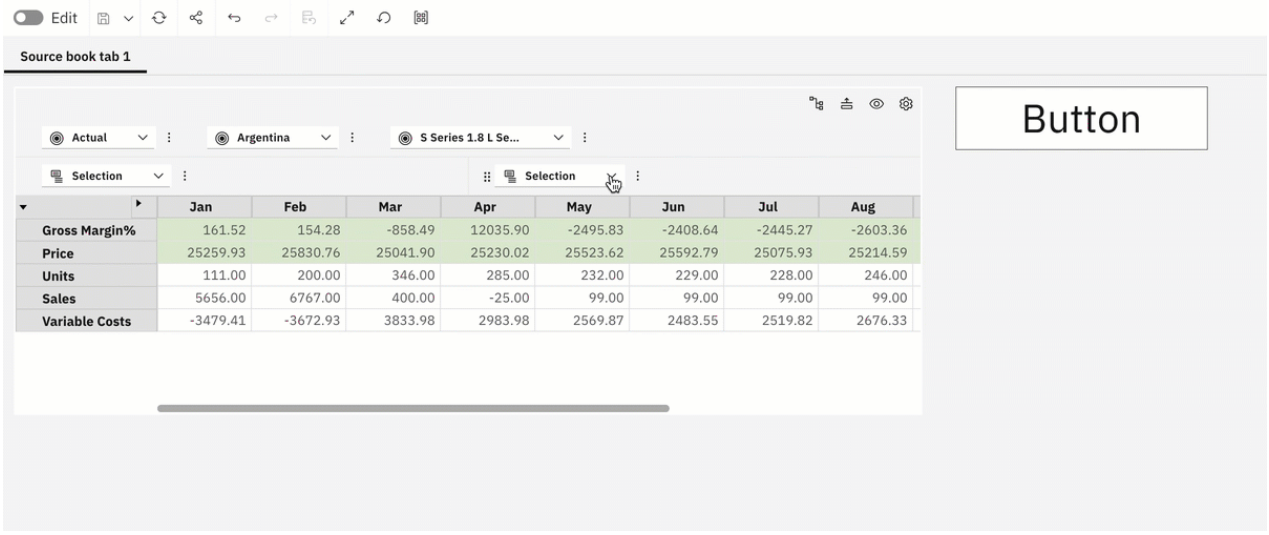
1. In edit mode, add an action button to the source book.
2. Go to the action button's **Properties** and under **Navigation**, enable **Navigate to destination**.
3. Click **Other book** and select a target book and the tab that you want to pass context to.
4. Click **Pass context**.

   The **Advanced context** link becomes available.
5. Click **Advanced context** and select **Custom** group mapping.

   **Note:** The **Default** group mapping displays the default synchronization groups and hierarchies. To edit or configure these settings, go to **Custom** group mapping.
6. Select the source group, which passes the context, the target group that receives the context, and the hierarchies you want to synchronize.

   You can add more mappings or remove existing ones under **Custom**.

Now, change the hierarchies in a widget in the source book and click the action button. You automatically navigate to the target book and hierarchies that were selected to synchronize, update in the widget.




## Run process

You can run a process using synchronization from the **Set parameters** dialog. Select the synchronization type (member or set) from the **Control Type** column for each parameter, then select the specific details for your synchronization type from the **Control Details** column.

**Remember:** Synchronize Default is the default synchronization group that you selected under the book's properties.

When you select a synchronization group, you can also select synchronized dimensions and hierarchies in the **Control Detail** menu. Only the synchronized dimensions and hierarchies in the selected group display in the **Control Detail** menu. Dimensions from control cubes are also supported in the **Control Detail** menu.

## Setting synchronization for objects

You can set synchronization for hierarchies, sandboxes, and certain gestures.

Before you begin, set the default synchronization group to be at a book level, sheet level, or custom group level.

1. In edit mode, with nothing selected, click **Properties**.
2. Under **Synchronization**, select the default synchronization group.
   - Book - widgets in all the sheets in a book synchronize
   - Sheet - widgets that are located in the same sheet synchronize
   - Custom group - all widgets in the book synchronize to the changes in the selected custom group

   **Note:** Sheet scope synchronization is the default.

   For information on custom groups and group level synchronization, see Group synchronization.

### Hierarchy synchronization

1. Select a view, visualization, or websheet, click **Properties**.
2. Select **Synchronize** and under **Hierarchies**, enable **Synchronize hierarchies** to enable synchronization.

   All hierarchies are selected by default. You can turn off synchronization for a hierarchy if you want to exclude it from the synchronization.

### Hierarchy synchronization across different databases

1. In edit mode, create a book with two views from cubes that are in two different databases.
2. Select a view and go to **Properties > Synchronize**.
3. Under **Hierarchies**, enable **Synchronize hierarchies** to synchronize all hierarchies.

   All hierarchies are selected by default. You can turn off synchronization for a hierarchy if you want to exclude it from the synchronization.
4. Repeat steps 2 and 3 for the second view.
5. Click **Properties** again to close the panel.
6. Click in the book and go to **Properties > Synchronization**.
7. Under **Mappings**, turn on **Synchronization across servers**.

### Gesture synchronization

1. Select the widgets that you want to synchronize and click **Properties**.
2. Select **Synchronize** and under **Hierarchies**, enable **Synchronize hierarchies** to enable synchronization.
3. Expand the **Gestures** section and enable **Synchronize gestures**.
   a. To keep or hide a row or column (including unhiding all), enable **Member keep / hide**.
   b. To expand or collapse a member, enable **Member expand / collapse**.

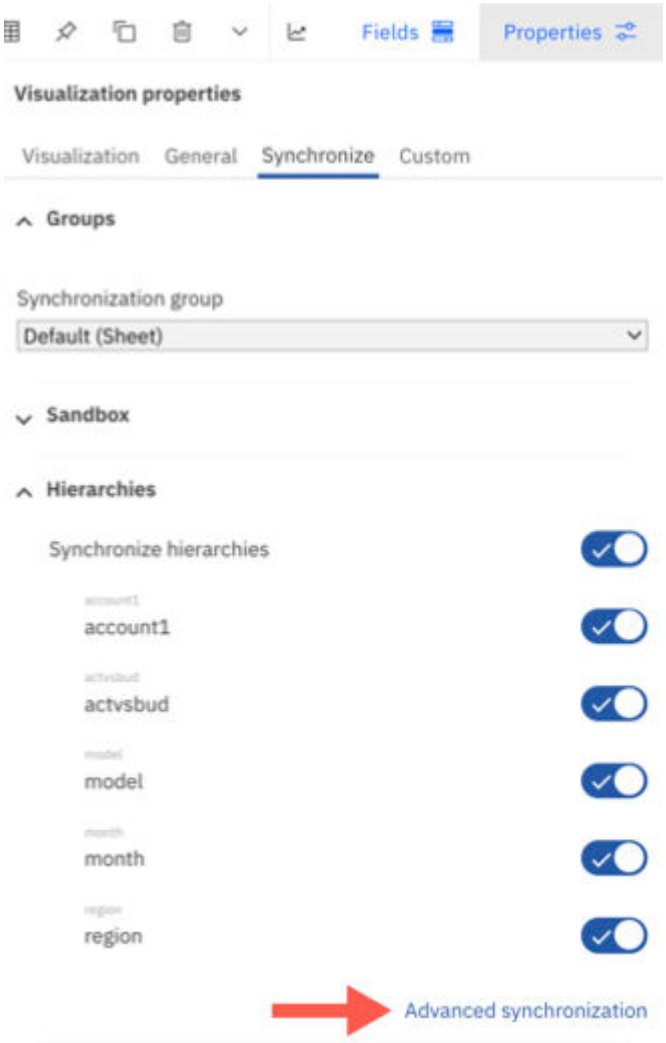
## Sandbox synchronization

1. Select a view, visualization, or websheet, click **Properties**.
2. Go to **Synchronize** and under **Sandbox,** turn on **Synchronize sandbox**.

## Advanced synchronization settings

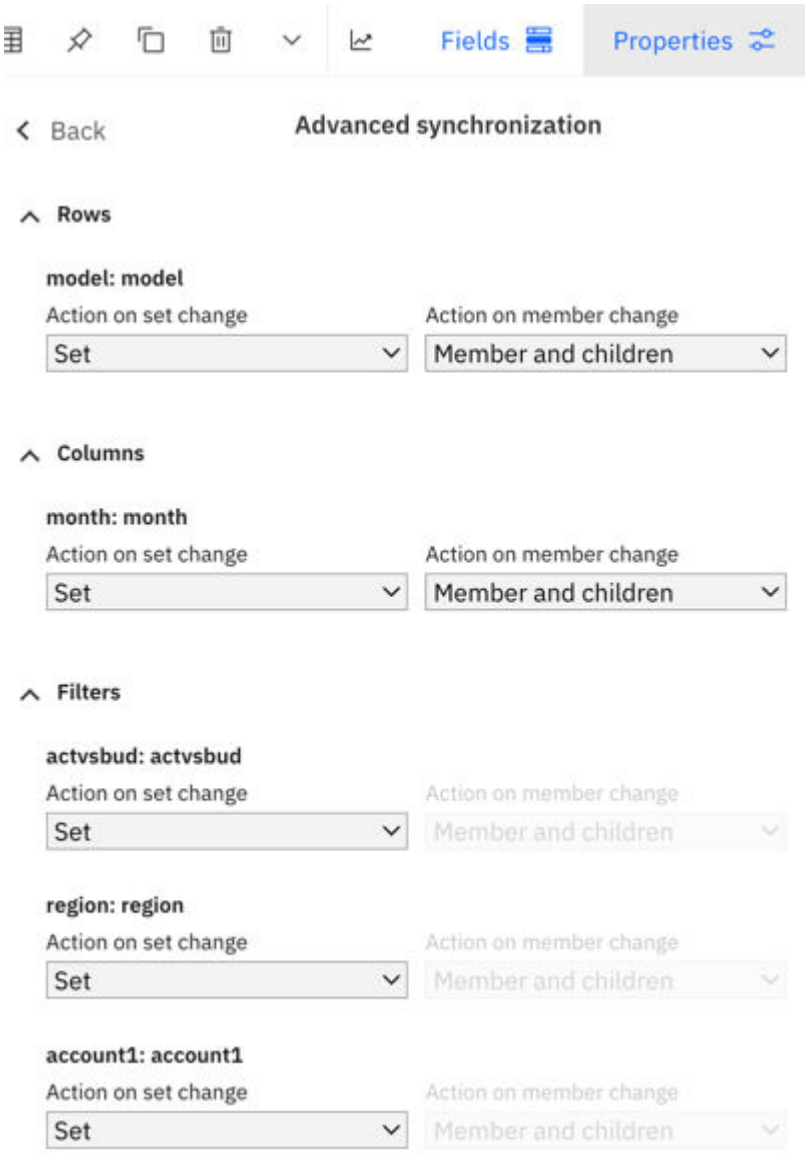
**Advanced synchronization** settings let you customize how each hierarchy in the cube viewer or visualization displays when a synchronization event is received by that widget.

You can find **Advanced synchronization** in the **Properties > Synchronize > Hierarchies** section when synchronization is enabled for the widget that receives the synchronization.




**Note:** The advanced synchronization settings apply to only the widget that receives the synchronization event.

The **Advanced synchronization** panel is divided into **Row**, **Column**, and **Filters** sections. You can choose what each hierarchy on the **Row** and **Column** areas displays when the widget receives a set change or member change synchronization event. You can select the hierarchy display for the **Context** area under **Filters**.




For a set change synchronization event, you can choose what you want the set in the receiving widget to display:

- Set - The set with all its members and children displays in the widget.
- Set and aggregate - The aggregate of the all the members in the set shows below the set.
- Aggregate and set - The aggregate of the all the members in the set shows above the set.
- Aggregate - Only the aggregate of all the members in the set displays in the widget.

Similarly, for a member synchronization event, you can choose what you want the set in the receiving widget to display:

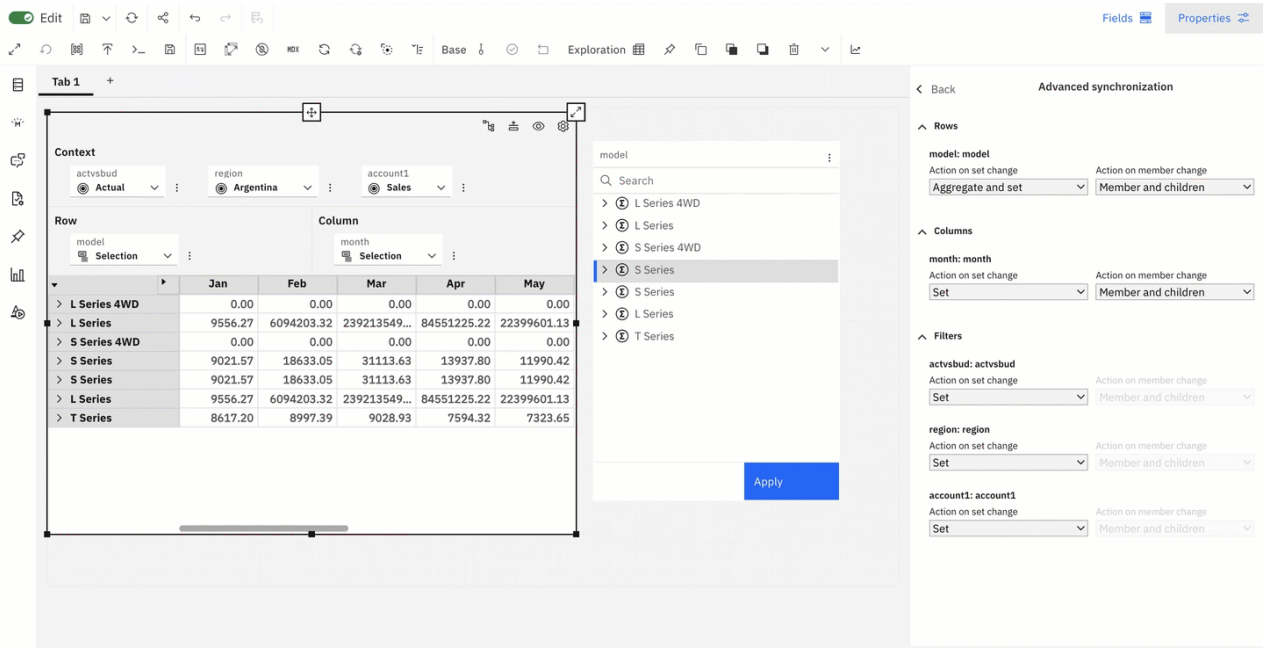
- Member and children
- Member
- Children
- Member and descendants
- Descendants
- Member and parents

- Parents
- Member and ancestors
- Ancestors
- Member and leaves
- Leaves

The advanced synchronization settings persist if you move a hierarchy from to a different area in the widget.

If you're using the selector widget to synchronize members, you can select multiple members and the receiving widget will display the set and members based on the synchronization settings you selected for that widget.

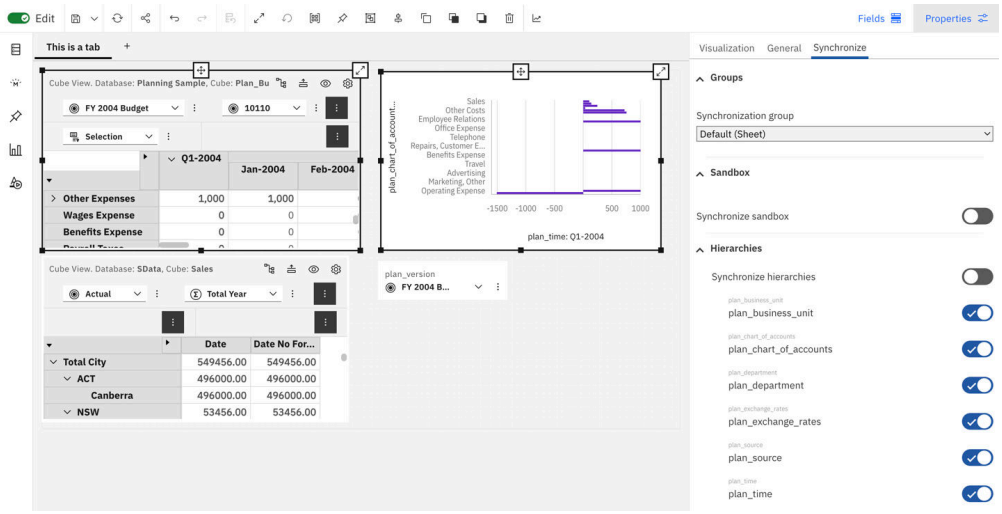
In the following example, advanced synchronization is set to **Aggregate and set** and **Member and children**for the *model* hierarchy on the **Row**. The selector list for *model* is used to synchronize members. When the member is changed to *L Series*, the cube viewer updates to display an aggregate of all the members in the set and the set updates to display only the selected member and its children.




## Set synchronization properties for multiple widgets simultaneously

You can set synchronization properties for multiple widgets at the same time.

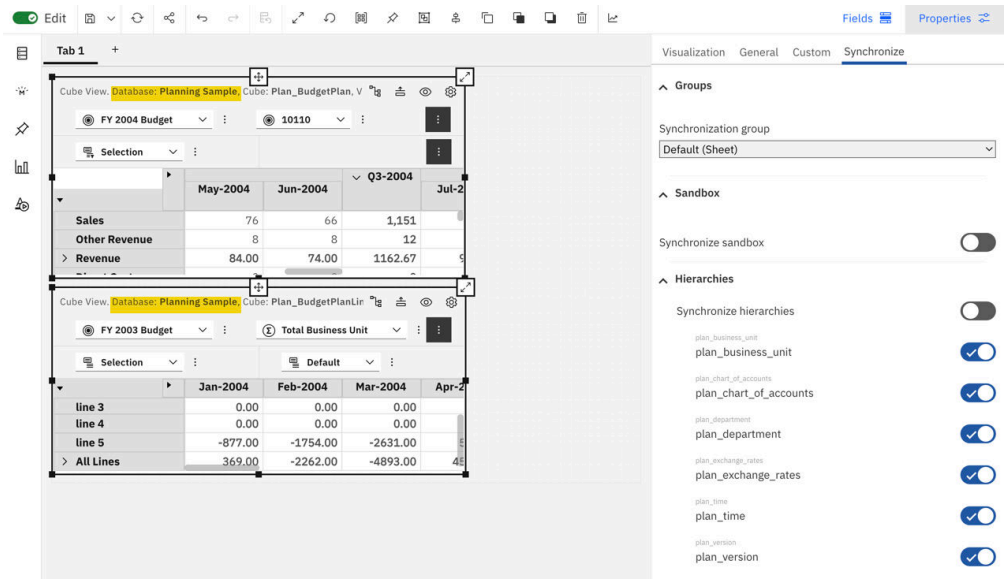
In **Edit** mode, select multiple widgets with Ctrl + click or by using the mouse drag action. Then, under **Properties > Synchronization**, choose the synchronization properties that you want to apply to the selected widgets.




When you select multiple widgets, only the synchronization properties that apply to all selected widgets display. The following properties are available if you select:

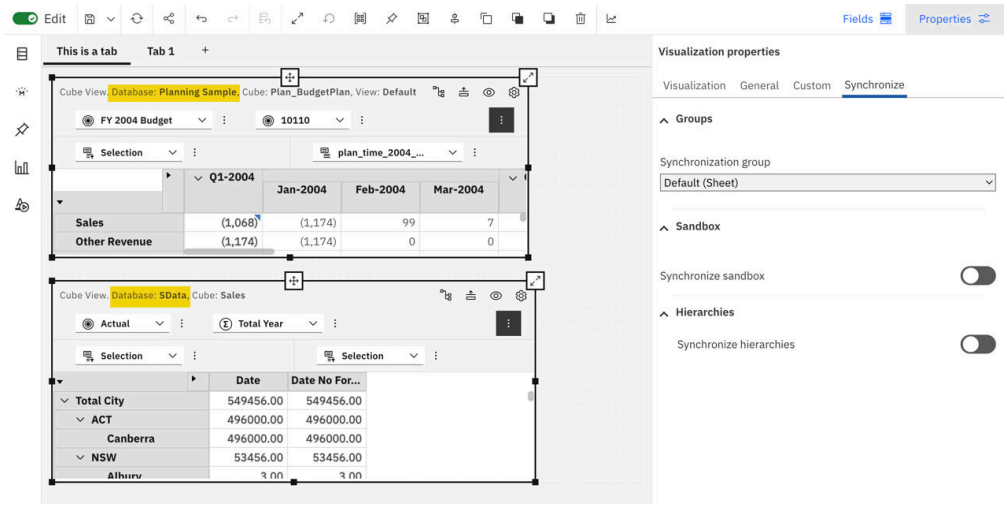
- Cube views from the same database

  The **Synchronization group**, **Synchronize sandbox**, and **Synchronize hierarchies** properties are available. You can synchronize only those hierarchies that are common to both cube views; other hierarchies do not display.



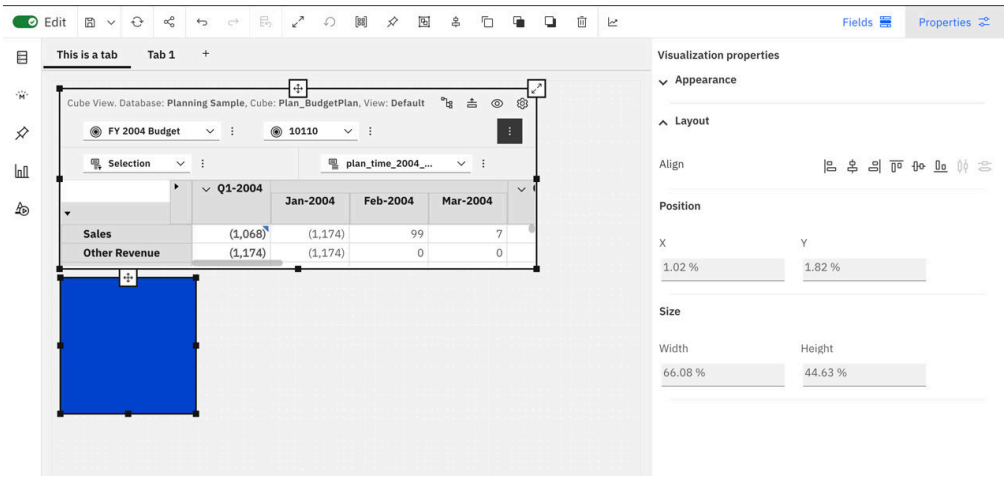

- Cube views from different databases

  The **Synchronization group** and **Synchronize sandbox** properties are available. You can also enable the **Synchronize hierarchies** property to turn on synchronization, but since there aren't any common hierarchies between the selected widgets, there are no hierarchies to select.




- Cube view and shape widget

  Since the shape widget does not have synchronization properties, no synchronization properties display.




The order in which the synchronization properties display in **Synchronization groups** and **Synchronize hierarchies** is determined by the widget that you select first. For example, if the first widget that you select is in the Book synchronization group, the Book option appears first in the **Synchronization groups** list.

# Data entry

You can enter data by typing in editable cells.

## About this task

Read-only cells are colored gray and cannot be updated directly with data entry. Calculated cells are cell values that are derived through rules calculation and cannot be directly edited. By default, cell values derived by rule calculation are colored green, but this depends on the cube view properties set by the modeler.

To enter dates, click in a date cell and select the date.

Consolidated cells are bold. You can spread data by typing a value in a consolidated cell if the **Consolidation TypeIn Spreading**, and **Allow Spreading** capabilities have been set to `Grant` by your administrator. These capabilities are set in the Architect or Perspectives Server Explorer. To find out more, see Capability assignments.

If any descendant leaf cells have a non null-value, a proportional spread is applied.

If all descendant leaf cells are null, an equal leaves spread is applied. An equal leaves spread distributes a specified value equally across the lowest child members of a consolidated cell.

You can use spreading shortcut keys. For example: S<>100 spreads the value 100 equally to all leaf cells of the consolidated cell, and replaces the existing leaf cell values. You can also use the spreading menu. To find out more, see “Apply data spreading” on page 611.

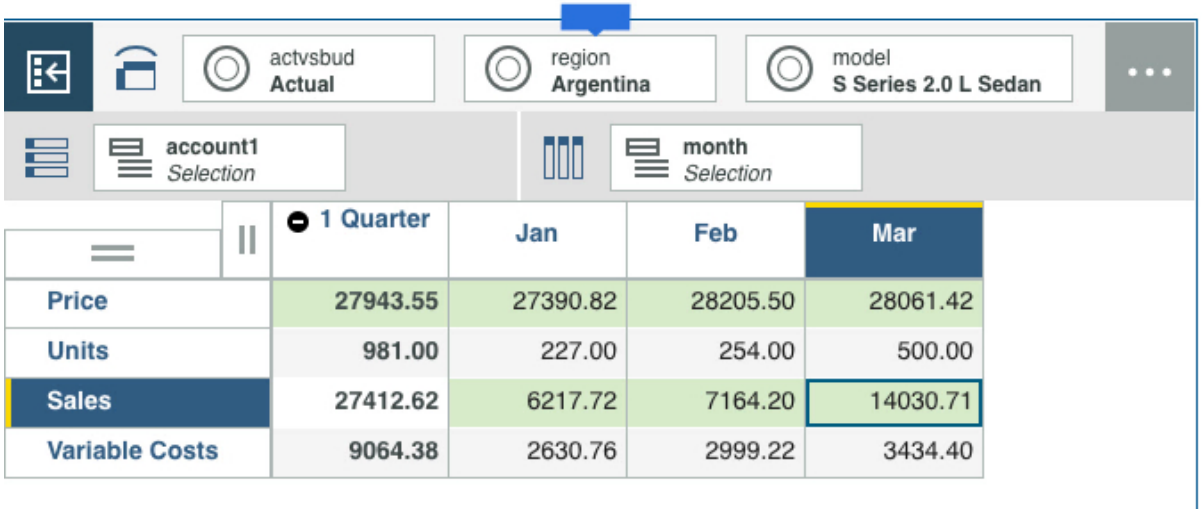
**Note:** Data in a view is displayed in accordance with your browser locale setting, using the decimal and thousands separators for your locale. You can double-click a cell to view the raw data value as it exists on the Planning Analytics database, which uses a period as the decimal separator and omits a thousands separator. The raw value may include more decimal places than are displayed in the cell.

When you enter data directly in a cell, use the decimal separator for your browser locale.

## Procedure

1. In the Price view, notice that the cells in the 1 Quarter column are in bold, so they are consolidated cells. Also, notice that some of the cells are shaded. These shaded cells can't be edited. For example, Sales is a calculated value based on Units and Price, so you can't edit values for Sales.
2. Click the cell at the intersection of Mar and Units. Type 500, and then click **Enter**.

   Notice that the value of Units under 1 Quarter reflects the change that you made. Notice also that the Sales values for Mar and 1 Quarter have also been updated.




3. Click the cell at the intersection of Feb and Units, then type 2K. 2K is a data entry shortcut for the value 2000. You can use all the shortcuts that are described in “Data spread keyboard shortcuts” on page 1254.
4. Right-click the cell at the intersection of Jan and Units, then select **Hold**.

   This places a hold on the cell, excluding it from any spreading actions. You can still edit the cell directly. Two vertical bars indicate that a hold is active on a cell 

.
5. Type S>700 in the cell at the intersection of 1 Quarter and Units.

   To get the same result, right-click in the cell and select **Spread data** > **Equal spread**, enter 700 in the **Value** field, and click **Apply**.

   Notice that 700 is evenly spread by placing the value 350 in the Feb and March children of 1 Quarter, but the value of Jan remains unchanged.
6. Right-click the cell at the intersection of 1 Quarter and Variable costs, then select **Hold**.

   When you place a hold on a consolidated cell, the consolidated value remains constant when you change any of the children. When a consolidation hold is in place and you change the value of a child, proportional spreading is automatically applied to the remaining children so that the consolidated value remains unchanged.
7. Type 2000 in the cell at the intersection of Mar and Variable Costs.

Notice that 1 Quarter remains unchanged, while Jan and Feb are updated to keep the consolidated value valid.

8. Right-click any cell, then select **Release all holds**. This removes all holds in the view. You can also right-click a cell with an active hold, then select **Release hold** to release a specific hold.

## Copy and paste

You can cut, copy, and paste values between views in Planning Analytics Workspace. You can also paste values from other applications, such as Microsoft Excel.

You can use the following keyboard shortcuts to cut, copy, and paste in a view:

| Action | Windows | Apple Mac |
|---|---|---|
| Cut | CTRL+X | CMD+X |
| Copy | CTRL+C | CMD+C |
| Paste | CTRL+V | CMD+V |

You can also right-click a cell or a range of cells, and select cut or copy. Paste must be done by using the keyboard shortcut.

When you paste data into a single cell, the paste expands to other cells depending on the number of cells that you copied.

**Note:** Not all browsers support the right-click menu, but all browsers support the keyboard shortcuts.

### Copy a single cell into multiple cells

You can copy a single cell and paste it into multiple cells.

1. Copy the cell.

   **Note:** To select a range of cells, hold down Shift and select the range of cells.
2. Click CTRL+V or CMD+V to paste.

### Copy a range of cells and pasting it into a larger range of cells

You can paste a range of cells into a larger range of cells, repeating the source range where possible. If you copy a range of four cells, the target range must be a multiple of four to repeat the cells. If you copy four cells into a target of nine cells, only the first four cells are pasted into. If you copy four cells into eight writeable target cells, the cells are repeated.

1. For a range of cells, hold down Shift and select the cells. Right-click the selection and select **Copy**.
2. Hold down Shift and select the target cells, then click CTRL+V or CMD+V to paste.

#### Copy two rows and four columns into two rows and eight columns

This example shows the results of copying two rows and four columns into two rows and eight columns. The header row is not copied.

The source cells that are copied:

| 1 Quarter | Jan | Feb | Mar |
|---|---|---|---|
| 90 | 25 | 30 | 35 |
| 87 | 28 | 30 | 32 |

The result of pasting the source cells into two rows and eight columns:

| 1 Quarter | Jan | Feb | Mar | 2 Quarter | Apr | May | Jun |
|---|---|---|---|---|---|---|---|
| 90 | 25 | 30 | 35 | 90 | 25 | 30 | 35 |
| 87 | 28 | 30 | 32 | 87 | 28 | 30 | 32 |

## Copy rows and columns

You can copy and paste whole rows or columns. You can select a single row or column, or a contiguous range of rows and columns.

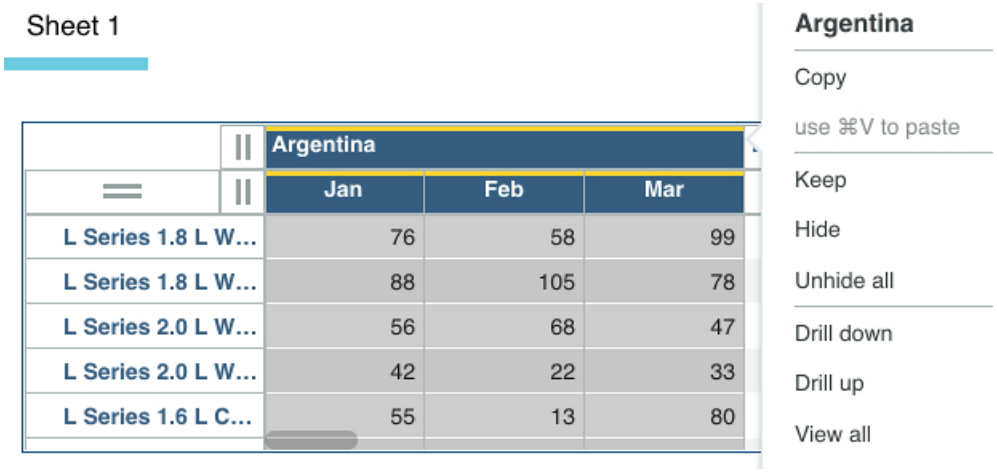
1. Select the row or column headers, then right-click the row or column headers, and select **Copy**.

   It can take a few seconds for data to be copied; a window appears that shows the progress.
2. When the data has been successfully copied, click **OK** in the window, then click the cell where the paste is to start.

   **Tip:** You can paste the rows or columns into a larger selection, repeating the data, by holding Shift and selecting the target rows and columns, in multiples of the source selection.
3. Click CTRL+V or CMD+V.

### Copy and paste from nested hierarchies by selecting rows or columns from the outer axis

You can copy and paste from nested hierarchies by selecting rows or columns from the outer axis. For example, suppose that you have Car models in the rows and Regions and Months nested on the columns. Select Argentina, this selects all the months underneath, and then copy. You can then paste the data into another nested area, such as Belgium.




### Copy cells from a column and paste them into rows

In a view without nesting on the rows and columns, you can select a range of cells in a column and paste them into multiple rows. For example, select and copy the values in the Argentina column for the L Series 1.8 L Wagon, L Series 1.8 L Convertible, and L Series 1.8 L Sedan rows. Then, select the cells for Belgium, Brazil, and Chile in the L Series 1.8 L Wagon row, and paste the cells into the selected cells.

| | Argentina | Belgium | Brazil | Chile |
|---|---|---|---|---|
| L Series 1.8 L Wagon | 76 | 0 | 0 | 0 |
| L Series 1.8 L Convertible | 97 | 0 | 0 | 0 |
| L Series 1.8 L Sedan | 56 | 0 | 0 | 0 |

The result is the selected values are copied into the corresponding cells for Belgium, Brazil, and Chile for the L Series 1.8 L Wagon, L Series 1.8 L Convertible, and L Series 1.8 L Sedan rows, as shown in this table:

| | **Argentina** | **Belgium** | **Brazil** | **Chile** |
|---|---|---|---|---|
| L Series 1.8 L Wagon | 76 | 76 | 76 | 76 |
| L Series 1.8 L Convertible | 97 | 97 | 97 | 97 |
| L Series 1.8 L Sedan | 56 | 56 | 56 | 56 |

### Copying data from Microsoft Excel

When you copy data from Microsoft Excel and paste it to Planning Analytics Workspace, you must use the same decimal locale separator in both applications. The locales in your browser and Windows must match.

Planning Analytics Workspace is a browser-based application and uses the browser locale setting. Microsoft Excel is a Microsoft Windows application, which uses the language setting in Windows and copies data to the clipboard by using the locale-specific decimal separator.

### Copy/paste and clear limitations

There is a 65,536 cell limit for copying and pasting into a view. An error message appears if you try to paste more. The same limit applies to clearing cells in a view.

Copy and paste is not supported into or from Websheets in Planning Analytics Workspace.

## Paste values to a mixed range of leaves and consolidated cells

You can paste values to leaves and consolidated levels in a view, if the paste special feature is enabled.

The paste special is not available by default. It is available on request only, and replaces the standard paste functionality.

### How paste special works

Cells that contain pasted values have the same values as the pasted values; they do not change. The values in leaves that are not pasted change to account for the distribution of the values.

Leaves are pasted first, and each cell is held after the update. Then, cells at the next level up (the first-level consolidation) are pasted to, and they too are held, and so on through the levels.

# Comments

You can add comments to cells to add information about the cell value.

You can add a comment to one cell at a time. You can't add comments to cells that don't display the original cell value. For example, if the **Show cell value** option is set to something other than **As is**, you can't add a comment.

## Use comments in Planning Analytics Workspace views

### Procedure

1. Right-click a cell, then click **Comments**.
2. Type your comment, then click **Add**.

You can add links to web pages in a comment. When the comment is posted, the link is displayed as a clickable link.

After you close the **Comments** window, comments are indicated by a blue triangle in the upper right corner of a cell.

3. To view comments in a cell, right-click the cell and select **Comments**.

   Comments are listed in descending order by date/time, with the most recent comment first in the list.

4. To delete a comment, select the comment in the **Comments** window, and then click 🗑. You can delete only your own comments. .

### Use comments in Planning Analytics Workspace Classic views

#### Procedure

1. Right-click in a cell and select **Comments**.
2. Click **Add comment**, type your comment, and then click **Post**.

   You can add links to web pages in a comment, for example `www.bbc.co.uk`. When the comment is posted, the link is displayed as a hyperlink, so when you click a link, it takes you to the web page.

   After you close the **Comments** window, comments are indicated by a blue triangle in the upper right corner of a cell.

3. To view comments in a cell, right-click the cell and select **Comments**.
4. Sort comments by user (who added the comment), and by date (the default) by clicking the **User** or **Date** heading. To sort on multiple columns, sort on one column, hold Shift, and then sort in the next column.
5. To delete a comment, select the comment in the **Comments** window, and then click **Delete**. You can delete only your own comments. To delete all of your comments, select the check box next to **Comment** and click **Delete**.

## Apply data spreading

IBM Planning Analytics Workspace provides a variety of pre-defined data spreading methods that you can use to distribute numeric data to cells in a cube view.

#### About this task

Predefining data spreading methods can be useful. For example, you can use data spreading to evenly distribute a value across a range of cells or to increment all values in a range of cells by a desired percentage.

When you use data spreading, there are several general steps to follow, which are applicable to all spreading methods.

**Step 1 - Select a cell or range from which you want to initiate data spreading:** You can initiate spreading from a single cell, a single linear range of cells, or a single rectangular range of cells. You cannot initiate spreading from a non-contiguous range of cells, nor can you spread data across multiple individually selected cells in a view.

**Step 2 - Select a data spreading method:** After you select the cell or range of cells from which you want to initiate data spreading, you can select a data spreading method. The Spread Data menu is dynamic; it displays only the methods that are valid for the cell or range from which you initiate spreading. For instance, the Relative Proportional Spread, Relative Percent Adjustment, Repeat Leaves, and Equal Spread Leaves methods are not valid for leaf cells. When you initiate spreading from a leaf cell, those data spreading method options appear dimmed on the Spread Data menu, indicating that they are unavailable.

**Step 3 - Specify required values:** With most spreading methods, you specify only the value you want to spread. However, a few spreading methods require additional values. For example, with the Straight

Line method, you specify both a Start Value and an End Value. With the Relative Proportional Spread and Relative Percent Adjustment methods, you must identify a reference cell for the spreading operation.

**Step 4 - Select the directions to extend spreading:** If you initiate spreading from a single cell, you must specify the direction(s) to extend spreading relative to the point of insertion. The cell from which you initiate data spreading is always included in the spread. Some data spreading methods allow you to extend spreading in multiple directions. These methods display **Direction** options as check boxes, of which you can choose any combination. When you initiate spreading from a selected range of cells, the **Direction** options are disabled and spreading is applied to the selected range.

**Step 5- Select an Update action:** The **Update action** indicates whether spread values should replace, be added to, or subtracted from existing cell values.

### Procedure

1. Right-click the cell or range of cells from which you want to initiate data spreading.
2. Click **Spread data options**.
3. Select a data spreading method.
4. Specify the required values and options.
5. Click **Apply**.

   You can also use shortcuts entered directly in cells to apply data spreading.

   See the following links to find out more about each data spreading method.

   - Proportional data spreading
   - Equal data spreading
   - Repeat data spreading
   - Clear data spreading
   - Percent change data spreading
   - Repeat leaves data spreading
   - Equal leaves data spreading
   - Straight line data spreading
   - Growth percent data spreading
   - Relative proportional data spreading
   - Relative percent adjustment data spreading

## Hide rows and columns

You can hide rows and columns that are not needed in a view.

### Procedure

1. Select the columns or rows that you want to hide, right-click, and select **Hide**.
2. Instead of hiding a member, you can select rows or columns that you want to keep, right-click, and select **Keep** or use the `keep` snap command. All other rows or columns are hidden.
3. To show columns or rows that are hidden, right-click and select **Unhide all**, or use the `unhide` snap command from the toolbar.

   For example, type `hide US, Feb-2004` in the snap command bar. See “Snap commands” on page 669.

## Set word wrap options in row and column headers

Word wrapping is on by default for column headers and off for row headers. You can also choose to break words, and clear word wrapping.

### Procedure

1. Right-click the row selector or the column selector and select **Word wrapping**.
   If you are using the new experience view in Planning Analytics Workspace, select **Text format** instead.
2. Select one of the options.

   **Word wrap**
   Word wrapping splits a long heading onto multiple lines. The breaks are made at the end of a word.

| Department Store | Direct Marketing | Equipment Rental Store | Eyewear Store |
|---|---|---|---|
| 7601.31 | 7601.31 | 7601.31 | 7601.31 |
| 619.32 | 619.32 | 619.32 | 619.32 |

   **Word break**
   Word breaking also splits a long heading onto multiple lines. Breaks can happen in the middle of a word.

| Department Store | Direct Market ing | Equipment Re ntal Store | Eyewear Stor e |
|---|---|---|---|
| 7601.31 | 7601.31 | 7601.31 | 7601.31 |
| 619.32 | 619.32 | 619.32 | 619.32 |

   **Clear wrapping**
   If word wrapping is cleared, headings that are too long to fit the row or column are truncated.

| Department... | Direct Mark... | Equipment ... | Eyewear St... |
|---|---|---|---|
| 7601.31 | 7601.31 | 7601.31 | 7601.31 |
| 619.32 | 619.32 | 619.32 | 619.32 |

   **Note:** If you are using the new experience view, clear word wrapping by selecting **Clear wrapping** under **Clear**.
3. After applying the **Word wrap** or **Word break** option, resize the row or column header height to accommodate the heading text.

## Make asymmetric selections on rows and columns

You can make asymmetric selections on rows and columns.

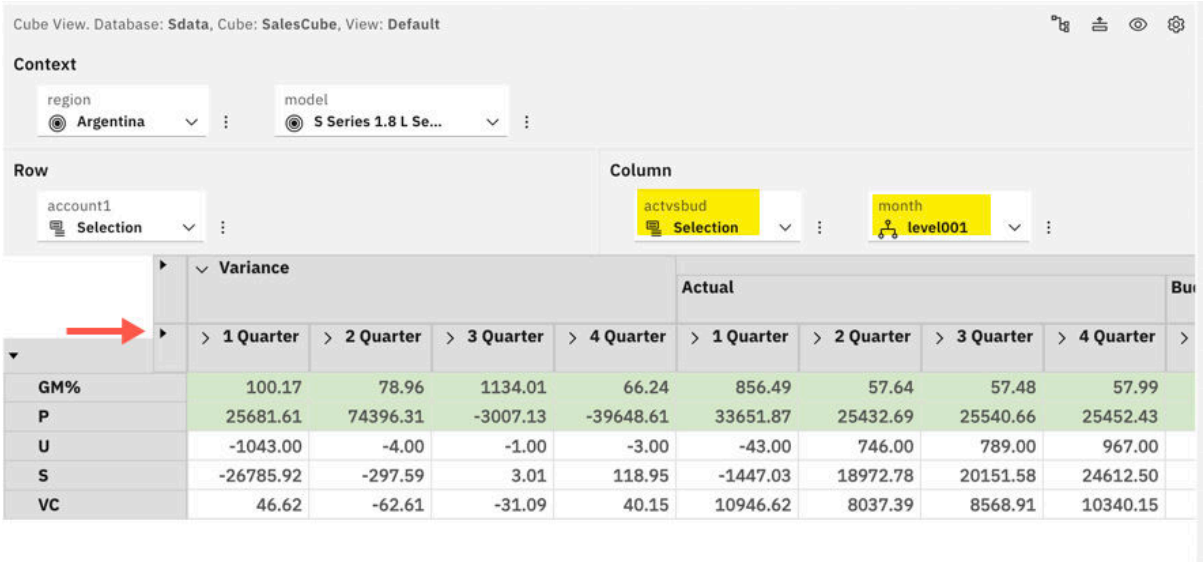
Suppose that you are looking at a view of sales, and you want to see the budget for the fourth quarter. Alongside the budget, you want to see actuals for the first, second, and third quarters. You can do this by making an asymmetric selection.

The following scenario uses the SData database.

### Making an asymmetric selection in Planning Analytics Workspace

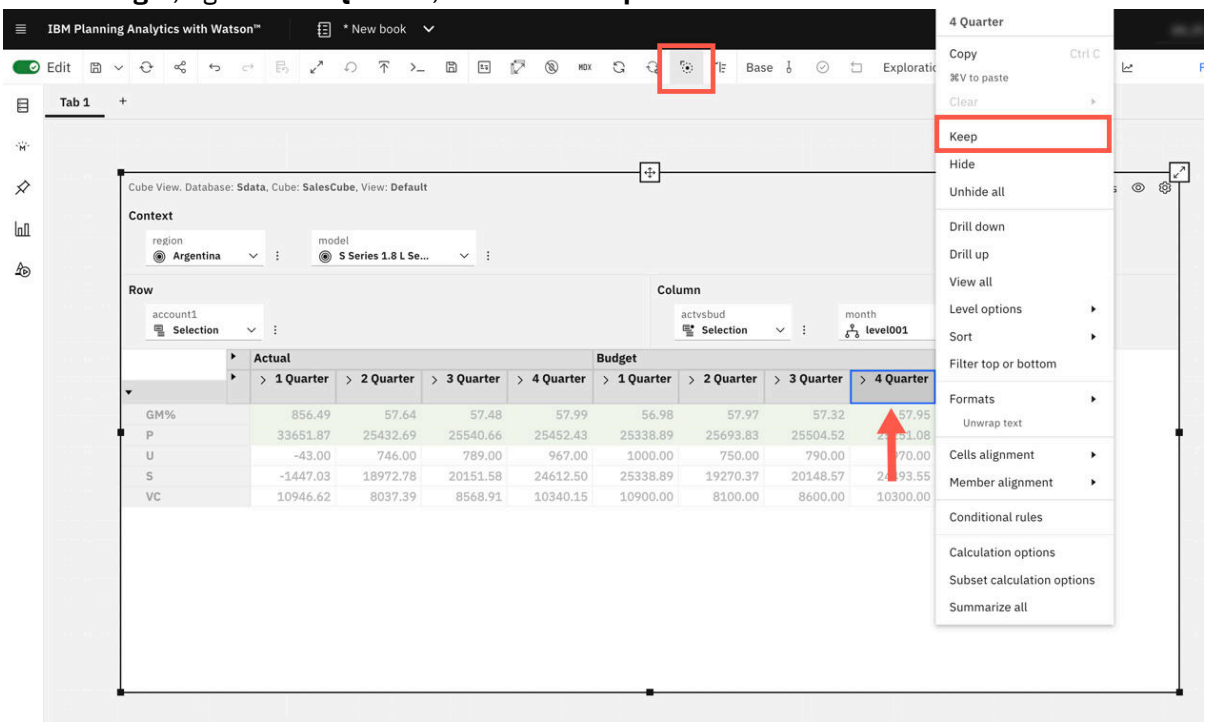
1. In a new book, go to the **SData** database, expand **Cubes** > **SalesCube** > **Views**, and drag the **Default** view onto the sheet.
2. Drag **actvsbud** from the Context area and drop it in front of **month** in the Column area.
3. Click **month** and under **Levels**, select **level001**.

You can now see the quarters.



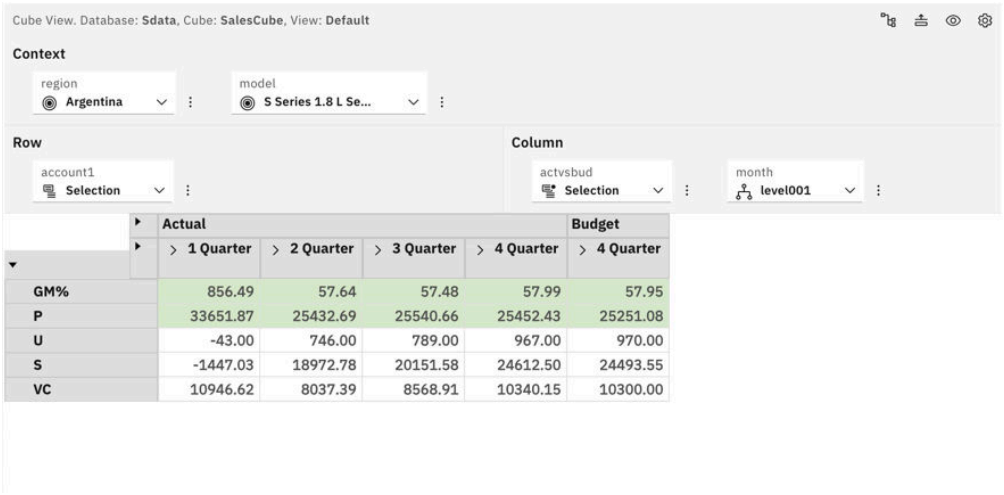

4. In the view, right-click the **Variance** heading, and click **Hide**.

   You can now see the **Actual** and **Budget** for the four quarters.
5. To make an asymmetric selection, click **Target selection** in the toolbar or press **F7**.
6. Under **Budget**, right-click **4 Quarter**, and select **Keep**.



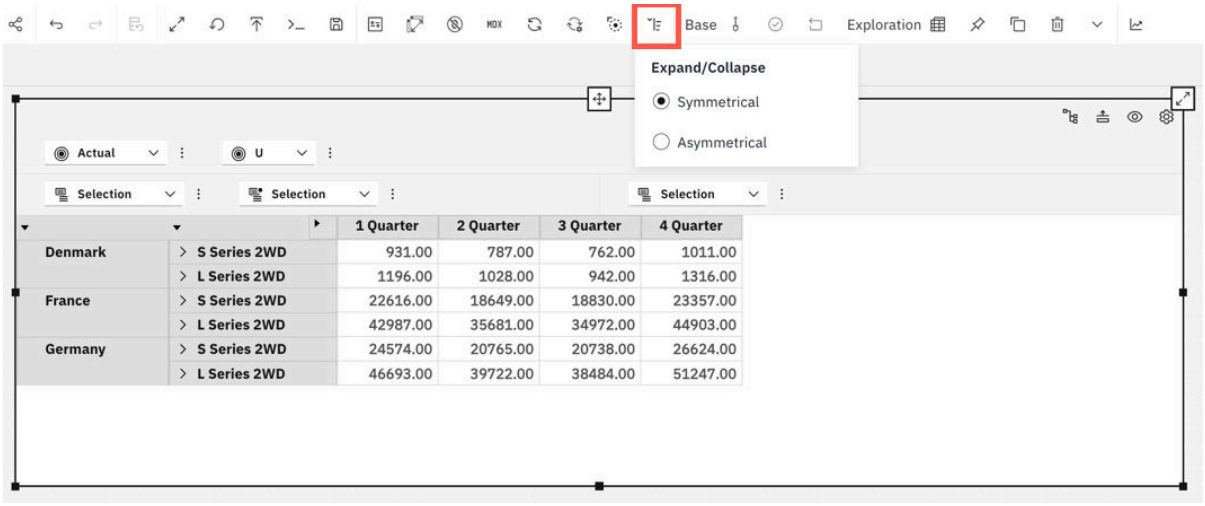

**Note:** To select multiple columns, click **Ctrl** while you click each member that you want to keep or hide in your asymmetric view. You can select only columns or rows that are visible in the view.

You can now see your budget for 4 Quarter, and your actuals for 1 Quarter, 2 Quarter, 3 Quarter, and 4 Quarter.

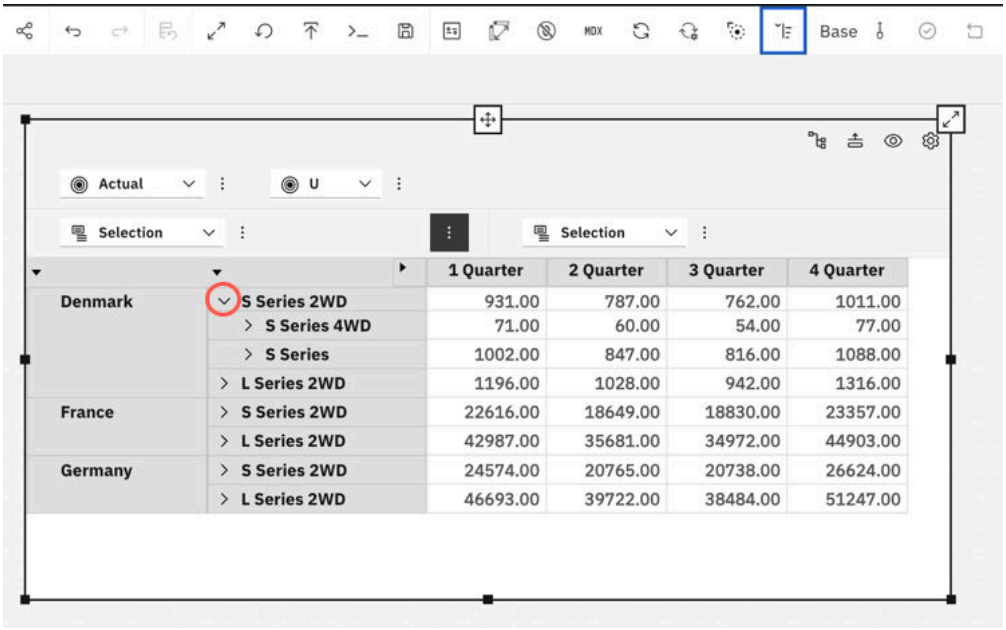



## Expand/collapse rows and columns asymmetrically or symmetrically

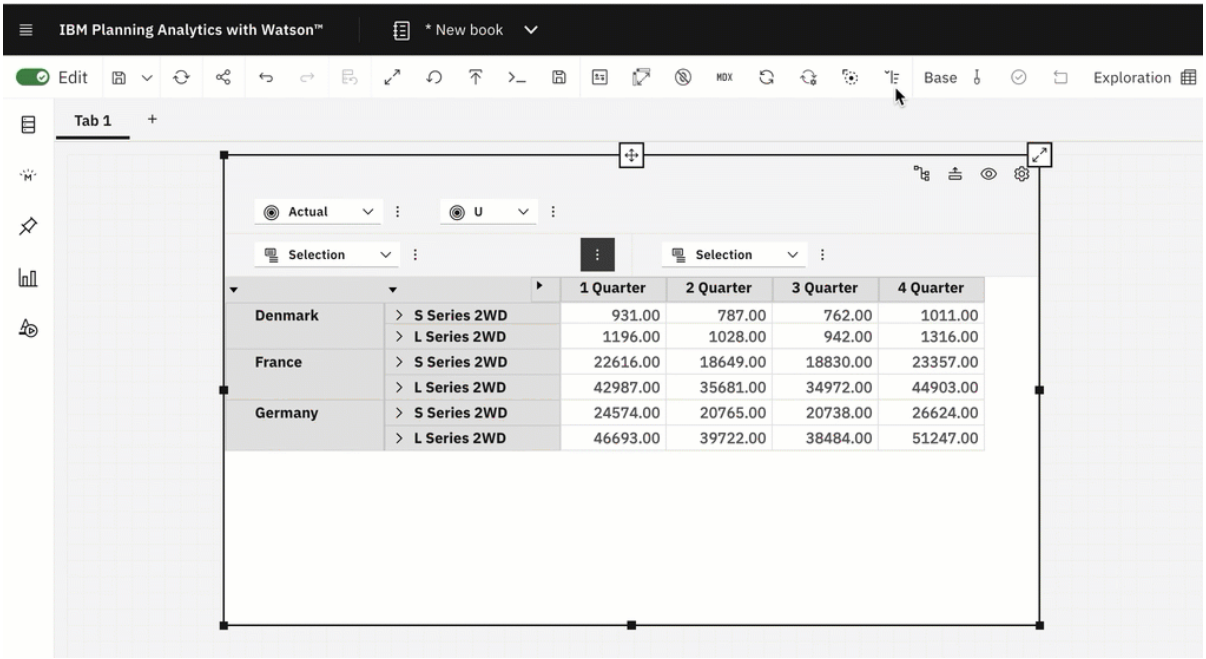
You can expand and collapse rows and columns in a view either asymmetrically or symmetrically. Click the **Expand/Collapse** icon in the toolbar and select how you want to expand and collapse rows and columns.




By default, expansion and collapse of rows and columns in views is asymmetrical. This means you can expand or collapse nested members individually.




In symmetrical expansion, expanding a nested member automatically expands that member across all hierarchies. In the following example, notice how expanding `S Series 2WD` for `Denmark` expands `S Series 2WD` for all countries.




**Tip:** If you set **Expand/Collapse** to be symmetrical but need to make an asymmetric gesture, you can do so with **Target selection**. Click **Target selection** or press **F7**, and expand or collapse an individual nested member.

# Lock row header column widths

You can lock the width of row header columns in Planning Analytics Workspace with the **Automatic row header width** option.

## About this task

The **Automatic row header width** prevents the row header columns from changing widths when you expand or collapse consolidation on the rows. The width is locked for any hierarchies or row header columns that are added to the report.

The **Automatic row header width** option is turned on by default.

## Procedure

1. Click **Properties**.
2. Navigate to the **Custom** tab.
3. In the **Exploration features** section,enable **Automatic row header width**.

# Drill down

You can drill down on a consolidated member in a view to view the members of the consolidation and to hide other members by double-clicking the member.

You can also drill down on members by using snap commands  from the toolbar. For example, type `Drill Georgia` in the snap command bar. See "Snap commands" on page 669.

## Procedure

1. To drill up to next level, right-click and select **Drill up**
2. To remove the drills, right-click and select **View all**, or use the `drill reset` snap command.

# Drill through to detailed data

You can click a cell in a cube view and drill through to detailed data in another view to get more detail or information about the selected cell. You can then add both views in the same sheet.

The drill-through view can either be a cube view, or an extract from a relational database.

You can drill through multiple levels of cube views, and add the drilled through views to the sheet.

**Note:** You can only drill to one level when the target is a relational database.

A drill process and drill assignment rule must be defined for a cell before you can drill through to detailed data. For more information, see "Configure drill-through to reveal detailed data" on page 843.

**Note:** To successfully drill through to an ODBC source, your administrator must create and configure an IBM Satellite Connector connection.

## Procedure

1. To add a drill through view to a book, you must be in edit mode.

   You can preview a drill through view when not in edit mode.
2. Right-click the cell that you want to drill through from, and select **Drill through**.

   The drill through targets are listed.
3. Select the drill through target that you want to see.

   The drill through view opens in a separate view that you can move on the sheet so you can see it next to your original view.

4. To add the view to the sheet, click **Add to current sheet**. Click **Cancel** to close the view.
5. If there are drill through targets to cube views defined in the drilled view, you can drill-though to multiple levels, and you can add one or more views to the sheet.
6. To see which cube and TM1 database the view comes from, hover over this button  in the view.

## Sort rows and columns

You can sort by dimension member or by data values, in either ascending or descending order. You can choose to sort respecting the hierarchy, or to sort breaking the hierarchy.

You can sort either from the menu, or by using snap commands, see "Snap commands" on page 669.

### Procedure

1. To sort by member, right-click either the row selector or the column selector, and select either **Sort ascending** or **Sort descending**.

   If you are using the new experience view, select **Sort** to see sorting options.
2. To sort by value, select either **Sort ascending** or **Sort descending**.
3. To sort values within a hierarchy in the dimension, right-click on the hierarchy label, select **Sort hierarchical**, then choose your option.

   If you are using the new experience view, you can sort values within a hierarchy under **Sort**.

## Filter rows and columns based on cell value

You can filter rows and columns based on cell values in a cube in Planning Analytics Workspace.

You can filter rows and columns by count or by value.

You can use filters and sorting capabilities simultaneously. For more information about sorting, see Sort rows and columns.

**Note:** You can have only one **Filter by value** or **Filter by count** filter active at the same time. If you apply another filter of the same type to the same view, the first filter is removed.

### Filter by count

Filter by count to filter members in a row or column to show the top or bottom members of a specified count type.

To filter a row or column by count, follow these steps:

1. Right-click on a row or column header.
2. Select **Filter**, and click **Filter by count**.
3. In the dialog box, choose the criteria for the filter:
   a. Select if you want the filter to show the top or bottom members of your specified count type.
   b. Type the number of members that you want to see included in the filter results. This value must be numeric.
   c. Select the type of count that you want to filter for. You can choose **Items**, **Percent**, or **Sum**.
   d. If you want to add an additional set of criteria for the filter, click **Add**. You can add a total of two sets of criteria in each filter. Click the remove icon ⊖ to remove a set of criteria from the filter.
4. Click **Apply**. The filter is applied to the row or column.

   A row or column with a filter applied has the filter icon .

## Filter by value

Filter by value to filter the members in a row or column using a comparison operator for a selected value. The value can be a number or a string.

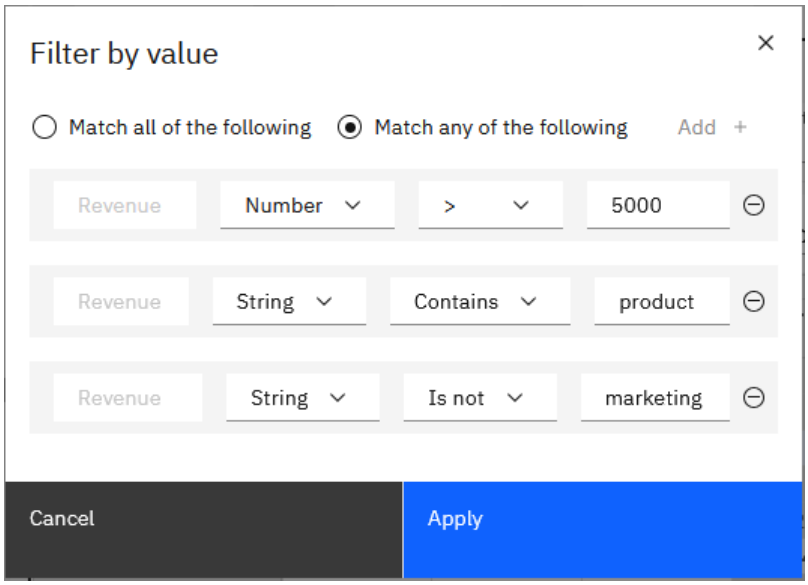
To filter a row or column by value, follow these steps:

1. Right-click a row or column header.
2. Select **Filter**, and click **Filter by value**.
3. In the **Filter by value** dialog box, specify the filter criteria.
   a. From the drop-down menu, select the data type: **Number** or **String**.
   b. Depending on the selected type, choose a comparison operator for the filter.

      For numeric values, choose from the following operators: **=** (equal to), **>** (greater than), **>=** (greater than or equal to), **<** (less than), or **<=** (less than or equal to).

      For string values, choose from the following operators: **Is** (equal to), **Is not** (is not equal to), **Contains**, **Starts with**, or **Ends with**.
   c. Type a number or string that you want the comparison operator to filter by.
   d. To add another set of criteria to the filter, click the **Add +** button, and repeat steps **a** to **c**.

      You can add a total of three sets of criteria that are based on only **Number** or only **String** values, or on both types of values, as shown in the following example:




      Click the remove icon to remove a set of criteria from the filter.
   e. Select how you want the filter to match the specified criteria.

      Select **Match all of the following** if you want the filter to include only members that match all of the criteria that you specified.

      Select **Match any of the following** if you want the filter to include members that match any of the criteria that you specified. You must select this option if your filter is based on both **Number** and **String** values.
4. Click **Apply**.

   The filter icon is applied to a row or column for which the filter is created.

## Editing an existing filter

To edit an existing filter, follow these steps:

1. Right-click on a row or column header that has the filter icon .
2. Select **Filter**, and click the filter that you want to edit.
3. Edit the existing criteria in the dialog box.
4. Click **Apply**.

### Clearing a filter

To clear a filter on a row or column, follow these steps:

1. Right-click on a row or column header.
2. Select **Clear**, and click **Filter**.

## Show and hide totals

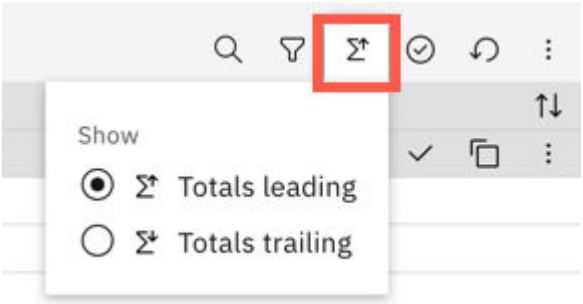
You can show and hide totals in a view, and choose whether totals are leading or trailing.

You can manage totals either from the row or column menu, from the Set Editor, or by using the `totals` snap command. To learn more, see "Snap commands" on page 669.

### Procedure

1. Right-click either the row selector or the column selector, and select one of the **Show totals** options.
2. When you edit a set, you can click the **Show Leading/Show Trailing** icon on the set editor to toggle the position of totals.

   **Note:** If you are using the new experience view and set editor, the **Show Leading/Show Trailing** icon looks slightly different.




## Filter on top or bottom members

You can filter to show only the top or bottom members in a view.

You might want to show the top three products by territory where the products are different for each territory. You can select how many members to show, and whether to base the sort on the member value, the percentile, or by sum.

### Procedure

1. Right-click, or tap and hold the row or column header, and select **Top or bottom filter**.

   **Note:** If you are using the new experience view, select **Filter top or bottom** instead.
2. Select whether you want to show top or bottom values.
3. Type the value that you want to filter on. The value depends on whether you are filtering on members, percentages, or the sum.

   - **Members**: Enter a number of top or bottom results. For example, you can type 50 to display the top 50 customers by revenue.

- **Percent**: Enter a percentage of top or bottom results. For example, you can type 10 to display the customers who contribute to the top 10% of revenue.
- **Sum**: Enter a number up to which your top or bottom results will sum. For example, you can type 10000000 to display the customers who contribute to the first 10 million dollars of revenue.

4. Select either **Members**, **Percent**, or **Sum**.
5. Click or tap **Apply**.
6. To clear the sort, right-click, or tap and hold the row or column header, and select **Clear filter**.

   To clear the filter in new experience views, right-click the row or column header and under **Clear**, click **Filter**.

## Suppress zeros

You can hide rows and columns in a view that contain only zeros.

You can suppress zeros in a view from the shortcut bar, by right-clicking on a row or column selector and selecting **Suppress zeros**, or by using the `zero` snap command.

### Procedure

1. To suppress zeros from the shortcut bar:
   a) Click anywhere on the view to display the shortcut bar handle.

      If you are using the new experience view, click anywhere on the view to select it.

   b) Click the handle, then click  .

      If you are using the new experience view, click **Suppress zeros** in the toolbar.

   c) Select a zero suppression option.

      **All**
      Suppress all rows and all columns that contain only zeros.

      **Rows**
      Suppress all rows that contain only zeros.

      **Columns**
      Suppress all columns that contain only zeros.

   d) Click **OK**.

   e) To unsuppress zeros, clear your selection.
2. To suppress zeros from a row or column selector:
   a) Right-click either the row selector or the column selector, then select **Suppress zeros**.

      You can also unsuppress zeros from this menu.

   b) You can suppress zeros on nested rows or columns by right-clicking the selector for the nested row or column and selecting **Suppress zeros**.

## Expand levels

You can expand levels in a hierarchy to a specific depth.

You can expand levels manually by clicking  , by selecting a specific level from the menu, or by using the `level` snap command. To learn more, see "Snap commands" on page 669.

### Procedure

1. To select a specific level to expand from the menu, right-click the member in the row or column that you want to expand.

   A member that can be expanded has this icon:  .
2. Click **Expand to level**, then select the level.

# Display cell values as percentages

You can change the way values are displayed so that values are displayed as percentages of the total.

## Procedure

1. Right-click in a cell, and select **Show cell value as**.
2. Select one of the following options:

   **% row total**

   Displays all the values in each row as a percentage of the total for the row.

   **% column total**

   Displays all the values in each column as a percentage of the total for the column.

   **% grand total**

   Displays values as a percentage of the total of all the values or data points in the report.

   **Advanced - % parent row total**
   Displays the values in each row as a percentage of the parent.

   **Advanced - % parent column total**
   Displays the values in each column as a percentage of the parent

   The cells that display a percentage are shaded. This shading indicates that the values in these cells are calculated.

3. To return to the actual cell values, in the **Show cell value** window, select **As-is**, or click  .

# Refresh data

You can refresh data for all views in a sheet or for individual views.

## Automatically refreshing data

You can set options to automatically refresh data. For details, see "Set data refresh options for a view" on page 623.

## Manually refreshing data in Planning Analytics Workspace

Click  on the book section of the toolbar to rebuild and refresh all views and websheets across all sheets in the current book.

To refresh an individual view, click anywhere in the view, then click  on the view section of the toolbar.

To refresh the data tree and ensure that you see the most recent views and other objects that are available to you, click  at the top of the data tree.

## Manually refreshing data in Planning Analytics Workspace Classic

On the dashboard, click  on the main toolbar, then click **Reload book** to rebuild and refresh all views and websheets across all sheets in the current book.

**Note:** For versions up to SC2.0.49, click  , then click **Reload book** .

To refresh all items on the current sheet, click  on the main toolbar, then click **Refresh book.**

To refresh an individual view, click anywhere in the view, then click  **Refresh**.

To refresh the data tree and ensure that you see the most recent views and other objects that are available to you, click  in the data tree. When you refresh the data tree, the tree collapses to the database level.

**Tip:** If you frequently work in an area of the data tree, you can bookmark the location so that you can quickly return to it after a refresh. For more information, see "Bookmark items" on page 672.

# Set data refresh options for a view

You can set data refresh options directly from the view toolbar. These options determine which actions in a view trigger a data refresh. Data refresh options for a view **do not** have any affect on how and when data is committed to the Planning Analytics server; they apply only to the refresh of the view as displayed in Planning Analytics Workspace.

## Set data refresh options for a view in Planning Analytics Workspace

### Procedure

1. In a book, click the view for which you want to set refresh options.
2. Click the **Refresh data** icon  on the view toolbar.
3. Set the data refresh option that you want to apply to the view.

   Select **After each context change** to refresh data after actions that change the structure of the view. This includes actions such as:

   - swapping rows and columns
   - moving hierarchies between rows, columns, context, and the bench
   - expanding or collapsing members in the rows or columns
   - using Snap commands

   If you clear this option, you must manually refresh data after modifying the structure of a view.

   **Note:** If your view structure changes as the result of a TurboIntegrator process execution, the view is not refreshed automatically when the **After each context change** option is enabled. A view that is modified by a process must be manually refreshed to see the resulting changes.

   **Defer on leaf data change** controls whether direct edits in leaf cells trigger a data refresh. When the option is selected, data is updated when the view is refreshed (either explicitly or by an action that triggers a refresh), and not when data is changed in leaf cells. Clear this option to refresh view data whenever a leaf cell is modified. This option is off by default when you create a new view in a book.

   Numeric data is always refreshed immediately in a view with any of the following actions:

   - data is entered in consolidated cells
   - data is spread in leaf cells
   - data is spread from the Spread data dialog
   - cells are updated from a date picker

# Rollback data entry

You can roll back data entries. Rolling back data entries is useful if you enter data into a cell, which triggers data spreading across a range of cells and you realize that you made the wrong change.

## Rollback data entry in Planning Analytics Workspace

### About this task

Rollback data entry does not undo any changes that are made to the book, such as swapping rows and columns. To undo book changes, click **Reset book**  .

**Note:** Rollback data entry does not apply to Websheets. Use **Undo**  to undo Websheet changes.

### Procedure

To roll back a data entry, click  in the toolbar.

Rollback undoes the last data entry that you made. To undo previous data entries, keep clicking  . The most recent entries are rolled back first. You can't redo a data entry rollback.

**Note:**

Sometimes you might not be able to roll back changes. For example, if you commit a sandbox, you can't roll back data entries. Another reason might be that the transaction log for the TM1 database is not enabled. Contact your TM1 administrator for help.

The transaction log is a record of everything that happens on the TM1 database. The log exists up until a **`Save data all`** action is performed on the TM1 database. You can roll back any data entries that are recorded by the transaction log.

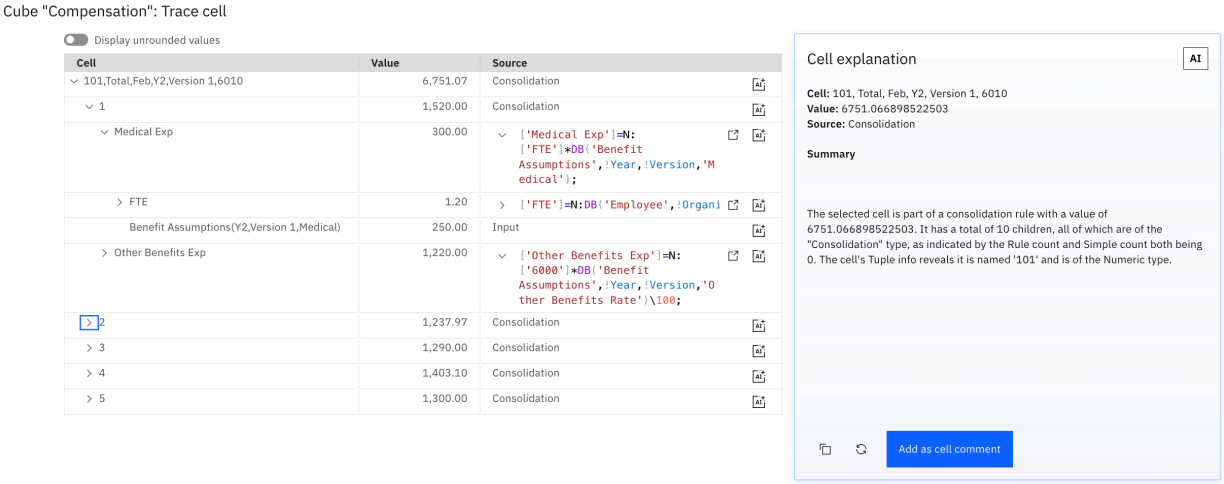
# Get cell explanation with AI

Use the **Explain cell** feature to get an explanation of the underlying values that contribute to a specific cell value. Cell values can be a result of complex calculations, several consolidations, or rules. The **Explain cell** feature uses generative AI to summarize the cell intersection, underlying logic of the cell value, and source type.

The initial release of the **Explain cell** feature is available only in Planning Analytics Workspace Cloud and in English only.

**Important:** The **Explain cell** feature is part of the Planning Analytics AI assistant add-on, which requires a separate license purchase; it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM® sales representative or go directly to your IBM account.

To get a cell explanation, right-click a cell in a cube view. Select **Explain cell [AI]** to generate the explanation.




The generated explanation includes a table, with the following columns, and an AI-generated summary:

**Cell**
: **Cell** identifies the original cell by its coordinates in the cube, by using a comma-separated list of member names. If there are members that contribute to the original value, you can drill down to view those members, their values, and source.

**Value**
: Displays the value of the original cell or member that contributes to the value of the original cell. If the value is over 10,000, the value that is displayed is truncated to an integer, otherwise the value shows two decimal places. You can hover over the displayed value to view the actual raw value as it exists in the TM1 database. To view the actual values for all data, enable **Display unrounded values**.

**Source**
: Identifies the source of the value: Consolidation, Input (a leaf cell that is not rule calculated), or rule. If the source is a rule, the entire rule is displayed. If the rule is too long to appear in its entirety within the confines of the column, click **Expand row** to reveal the full rule. You can also click the **AI** icon in each source row to generate an explanation for that member cell.

The explanation provides a descriptive summarization of the cell intersections along with underlying logic for the value held by the cell.

After you generate a cell explanation, you can add the summary to the related cell as a comment.

## AI-generated summary

The **Explain cell** feature uses granite-3-8b-instruct from the Granite family of IBM foundation models to generate the summary.

The Granite models are decoder-only models that can efficiently predict and generate language. These models were built with trusted data that has the following characteristics:

- Sourced from quality data sets in domains such as finance (SEC Filings), law (Free Law), technology (Stack Exchange), science (arXiv, DeepMind Mathematics), literature (Project Gutenberg (PG-19)), and more.
- Compliant with rigorous IBM data clearance and governance standards.
- Scrubbed of hate, abuse, and profanity, data duplication, and blocklisted URLs, among other things.

**Note:** IBM is committed to building AI that is open, trusted, targeted, and empowering. For more information about contractual protections that are related to IBM indemnification, see the IBM Client Relationship Agreement and IBM watsonx.ai service description.

# Open Planning Analytics for Microsoft Excel views

You can open and save views created in IBM Planning Analytics for Microsoft Excel if the view is saved to the Planning Analytics Workspace Content Store.

When views are saved to the Planning Analytics Workspace Content Store from Planning Analytics for Microsoft Excel, the view appears as a regular view in Planning Analytics Workspace. Edits and changes can be seen in Planning Analytics for Microsoft Excel after the view is saved in Planning Analytics Workspace.

You open the view from the **Welcome** page. Navigate to the folder where you saved it in Planning Analytics for Microsoft Excel. For example `Personal/My_views`.

**Note:** Permissions for the view are inherited from the folder that the view is saved in.

# Conditional formats

Most financial analysis focuses on highlighting exceptions in rows or columns of data, and not the whole table.

With conditional formatting, you can highlight specific values or cells, by changing what a cell looks like based on a set of conditions that you select. For example, you might want to highlight cells in S Series models that have higher sales than L Series models.

You can apply conditional formatting to both numeric and text values, and you can highlight empty cells by using conditional formatting.

Conditional formatting can also be applied to cell values that have been added to the sheet.

**Note:** You cannot add conditional formatting on the Apple iPad.

## Procedure

1. Ensure that you are in edit mode.

   **Note:** You can add conditional formats when you are not in edit mode, but you can't save them.
2. Right-click on a row or column header, or cell value that has been added to the sheet, and select **Conditional format**. If you are using the new experience view, select **Conditional rules**.

   In the **Conditional format** window, the member in the header that you selected is displayed. You can change this selection by clicking the member name and selecting a new member.

   a) Choose one of the operators that you want to apply.

   For numeric values, you can choose one of the following options:

   **<**
   Less than

   **>**
   Greater than

   **=**
   Equal

   **<>**
   Not equal to

   **>=**
   Greater than or equal to

   **<=**
   Less than or equal to

   For text values, you can select one of the following options:

   - Equals
   - Contains

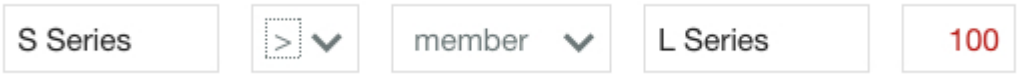
- Starts with
- Ends with
- Is empty

b) Select whether you want to compare the first member to another member, to a value, or to text.

c) Select the member, enter the value, or enter the text that you want to compare the first member to. If you want to highlight empty text cells, leave this cell empty.

3. Click and select what you want the cell to look like. You can select cell format (cell color and border color), font color or style, and icon sets.

   A preview of the formatting is shown. For example:




4. Click **Apply**.

   **Note:** Once applied, you can change the conditional formatting colors from **Properties**. Go to **Properties**, and under **Visualization**, click **Change color palette** to change the colors.

5. To add more conditional formats, click ⊕.
6. Reorder the conditional formats by using the arrow keys. ↑↓

   The order of conditional formats determines which formatting is used when there are conflicts between formats.

   If you have a format on both a row and on a column, and there are conflicts between the formats, the column formatting applies. For example, suppose that you have a different color background in the row and column, the color selected for the column is used.
7. Click **Save**.
8. To clear formatting, right-click on the column heading and select **Conditional format** (or **Conditional rules**). Select the formats that you want to delete, and click .

## Use the keyboard to work with conditional formatting

You can work in the conditional formatting editor by using the keyboard instead of the mouse.

### Navigation and selection overview

**Move between enabled items**

Press Tab to move to the next item.

Press Shift+Tab to go to the previous item.

**Open a menu**

With the focus on the menu, and press Enter.

**Move through items in a menu and members**

Press the up and down arrow keys. To move to a sub menu, press the right-facing arrow key.

**Select an option**

Press Enter.

**Enter the formatting area from the conditional formatting editor**

The formatting area is where you set the cell format, the text format, and the icon set to be displayed by the conditional format.

Tab to and press Enter to display the formatting area. Then press Space or Enter to enter the formatting area.

**Move between tabbed screens**

Press Tab to move between **Cell Format**, **Text Format**, and **Icon Set**.

**Move through options in the formatting area**

Use the up and down arrow keys to move from one option to another. For example, to move from **Font Style** to **Font Color**.

**Save**

On a Windows keyboard, press End to focus on **Save**, then press Enter.

On an Apple Mac keyboard, press the up arrow key until you move out of the formatting area, and then tab until **Save** has the focus. Press Enter.

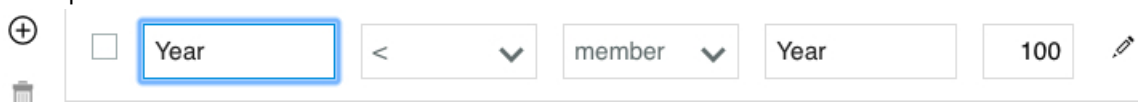
**Show or hide formatting area**

On a Windows keyboard, when focus is on either **Cell Format**, **Text Format**, or **Icon Set**, press the Home key to focus on , then press Enter.

## Add a new conditional format by using the keyboard

You can add and edit conditional formats in the conditional formatting editor by using the keyboard.

1. To create the first conditional format, in the editor, tab to the first member tile so that it has focus. The example shows the Year tile with focus.




   **Note:** To add another new conditional format, tab to **Add**, , and press Enter. A new conditional format line is added to the top of the list.

2. To change the member, press Enter to display the list of members. Scroll through the list with the up and down arrow keys, and select a new member by pressing Enter.
3. To change the operator from **<** (less than), tab to the operator selection, press the down arrow key to change the operators, and scroll through the list with the up and down arrow keys. Select a new operator by pressing Enter.
4. To change what you are comparing the first member to from **member** to **value** or **text**, tab to the **member** tile, and press the down arrow key. Scroll through the list with the up and down arrow keys, and select **member**, **value**, or **text** by pressing Enter.
5. If you selected member, to change the member value, tab to the next box and press Enter to display the list of members. Scroll through the list with the up and down arrow keys, and select a new member by pressing Enter.
6. If you selected **value**, or **text**, tab to the empty box and type the appropriate value or text string.
7. To select the condition formatting to be applied, tab to and press Enter.
   a. Move between the **Cell Format**, **Text Format**, and **Icon Set** tabs by pressing the tab key.
   b. Press Space, or Enter to go into the formatting area. To move between options such as **Fill Color** and **Border Color**, use the up and down arrow keys.
   c. To select a format, such as **Fill Color**, tab to the color. To move to **Border Color**, press the down arrow key until **Border Color** has focus, and tab to the color.




8. To save, on a Windows keyboard, press End to focus on **Save**, then press Enter. On an Apple Mac keyboard, press the up arrow key to move out of the formatting area, then tab to **Save** and press Enter.

# Calculations

Add calculations to a view to enhance your analysis. Calculations are saved as part of the view in the Planning Analytics Workspace content store.

You can create two types of calculations:

- **Summary calculations** apply to all visible leaf members and unexpanded consolidated members on a row or column.
- **Member calculations** apply to selected members on a row or column.

## Add a summary calculation to a view

A summary calculation applies to all leaf members and all unexpanded consolidated members that are visible on a row or column. The calculation updates dynamically as you change the members visible in the view, by drilling down or rolling up consolidations, or by changing the set used in the view.

### About this task

You can create summary calculations in Exploration (table) visualizations.

### Procedure

1. Right-click a row or column label and click **Summarize all**.

   Optionally, you can type a name for the calculation.
2. Click the calculation that you want to create.

   **Average**
   Displays the average value of the members.

   **Minimum**
   Displays the smallest value of the members.

   **Maximum**
   Displays the largest value of the members.

   **Median**
   Displays the median value of the members (the middle value). If there are an even number of values, it displays the average of the two middle values.

   **Sum**
   Displays the total of the members.

   **Aggregate**
   An aggregate calculation allows you to summarize values that can't simply be added up or averaged. For example, percentage or ratio values that are the result of a rule calculation.

   Aggregate displays the aggregate value of the members. This value is affected by whether a member has a C: level rule (a rule that applies to consolidated cells) associated with it or not. If the member has no C: level rule associated with it, Aggregate displays the sum. If the member has a C: level rule, the resulting value is the same as the consolidated member aggregating the same members. To use this feature, you must have IBM Planning Analytics version 2.0.9 or later.
3. Click **OK**.

#### Maximum and Sum

This example shows the result of applying the **Maximum** and **Sum** calculations to a view that contains a combination of leaf members, expanded consolidated members, and unexpanded consolidated members.

| | Europe | Scandinavia | Benelux | Belgium | Luxemburg | Netherlands | Maximum (all) | Sum (all) |
|---|---|---|---|---|---|---|---|---|
| Year | 46821 | 1808 | 2807 | 1270 | 128 | 1409 | 1808 | 4615 |
| 1 Quarter | 5629 | 137 | 336 | 149 | 16 | 171 | 171 | 474 |
| Jan | 5623 | 137 | 336 | 149 | 16 | 171 | 171 | 473 |
| Feb | 3 | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

Here, Europe and Benelux are expanded consolidated members. The values for these members are excluded from the summary calculations.

Scandinavia is an unexpanded consolidated member, so its value is included in the summary calculation, as are the leaf members Belgium, Luxemburg, and Netherlands.

The Maximum calculation for Year returns 1808, the value for Scandinavia. Though both 46821 (Europe) and 2807 (Benelux) are greater than 1808, these values are excluded from the calculation because they are expanded consolidations.

Similarly, the Sum calculation for Year returns 4615, which is the sum of Scandinavia + Belgium + Luxemburg + Netherlands. (1808 + 1270 + 128 + 1409 = 4615) Both Europe and Benelux, the expanded consolidated members, are excluded from the calculation.

## Member calculations

Member calculations apply to selected members on a row or column.

If a single member is selected, you can choose from the following calculations.

*Table 4. Single member calculation options*

| Calculation name | Description |
|---|---|
| Rank | Ranks the values of the selected member. Rank 1 is the highest value.<br>Rank is applied to the values visible in the view. If you drill down or roll up consolidations in the view, or if you modify the set used in the view, rankings change to reflect the ordering of the values that are visible in the view.<br>If you attempt to rank a mixture of string and numeric values, the string values are assigned a rank equivalent to a NULL numeric value. String values will always be ranked below any numeric values. |
| Absolute value | Displays the absolute value for the selected member. |
| % of total | Displays the member value expressed as a percentage of the root level consolidation (total). |
| % of parent | Displays the member value expressed as a percentage of its immediate parent. |
| Member Name + | Enter a numeric value in the entry box.<br>Displays the sum of the member value and the specified numeric value. |
| Member Name - | Enter a numeric value in the entry box.<br>Displays the difference of the member value minus the specified numeric value.<br>You can click to reverse the order of the values, subtracting the member value from the specified numeric value. |

| *Table 4. Single member calculation options (continued)* | |
|---|---|
| **Calculation name** | **Description** |
| Member Name / | Enter a numeric value in the entry box.<br>Displays the quotient of the member value divided by the specified numeric value.<br>You can click  to reverse the order of the values, dividing the specified numeric value by the member value. |
| Member Name * | Enter a numeric value in the entry box.<br>Displays the product of the member value and the specified numeric value. |

If two members are selected, you can choose from these calculations.

| *Table 5. Two members calculation options* | |
|---|---|
| **Calculation name** | **Description** |
| Average | Displays the average of the two member values. |
| Minimum | Displays the smaller of the two member values. |
| Maximum | Displays the larger of the two member values. |
| Median | Displays the median of the two member values. |
| Aggregate | Aggregate displays the aggregate value of the members. This value is affected by whether a member has a C: level rule (a rule that applies to consolidated cells) associated with it or not. If the member has no C: level rule associated with it, Aggregate displays the sum. If the member has a C: level rule, the resulting value is the same as the consolidated member aggregating the same members. To use this feature, you must have IBM Planning Analytics version 2.0.9 or later. |
| Member1 * Member2 | Displays the product of Member 1 multiplied by Member2. |
| Member1+ Member2 | Displays the sum of Member1 and Member2. |
| Member1 - Member2 | Displays the difference of Member1 minus Member2.<br>You can click  to reverse the order of the values, subtracting Member1 from Member2. |
| Member1 / Member2 | Displays the quotient of the Member1 divided by Member2.<br>You can click  to reverse the order of the values, dividing Member2 by Member1. |
| Member1 % change from Member2 | Displays the percent change from Member1 to Member2.<br>You can click  to reverse the order of the values, displaying the percent change from Member2 to Member1. |
| Member1 % of Member2 | Displays the value of Member1 expressed as a percentage of Member2.<br>You can click  to reverse the order of the values, displaying the value of Member2 as a percentage of Member1. |

If three or more members are selected, you can choose from these calculations.

| *Table 6. Three or more members calculation options* | |
|---|---|
| **Calculation name** | **Description** |
| Average | Displays the average all selected member values. |
| Minimum | Displays the smallest all selected member values. |
| Maximum | Displays the largest of all selected member values. |
| Median | Displays the median of all selected member values. |
| Sum | Displays the sum of all selected member values. |
| Aggregate | Aggregate displays the aggregate value of the members. This value is affected by whether a member has a C: level rule (a rule that applies to consolidated cells) associated with it or not. If the member has no C: level rule associated with it, Aggregate displays the sum. If the member has a C: level rule, the resulting value is the same as the consolidated member aggregating the same members. To use this feature, you must have IBM Planning Analytics version 2.0.9 or later. |

# Add a member calculation to a view

Member calculations apply to one or more members on either the row or column axis of a view.

## About this task

You can only create a calculation in an Exploration (table) visualization.

The calculations available vary according to the number of members selected. On a mobile device, you can only select a single member for a calculation.

## Procedure

1. Select and right-click the row or column label of the members for which you want to create a calculation and click **Create calculation...**.

   **Tip:** On a mobile device, click the row or column label, then click  .
2. Optionally, enter a name for the calculation.
3. Click the calculation you want to create, see “Member calculations” on page 630.
4. If you are creating an arithmetic calculation on a single member, enter the required numeric value.
5. Click **OK**.

# Add an asymmetric calculation to a view

You can create calculations that apply to nested dimension sets on rows or columns in a view. Known as asymmetric calculations, these calculations are applied to specific selected members and are not common to all nested sets for the dimension.

## About this task

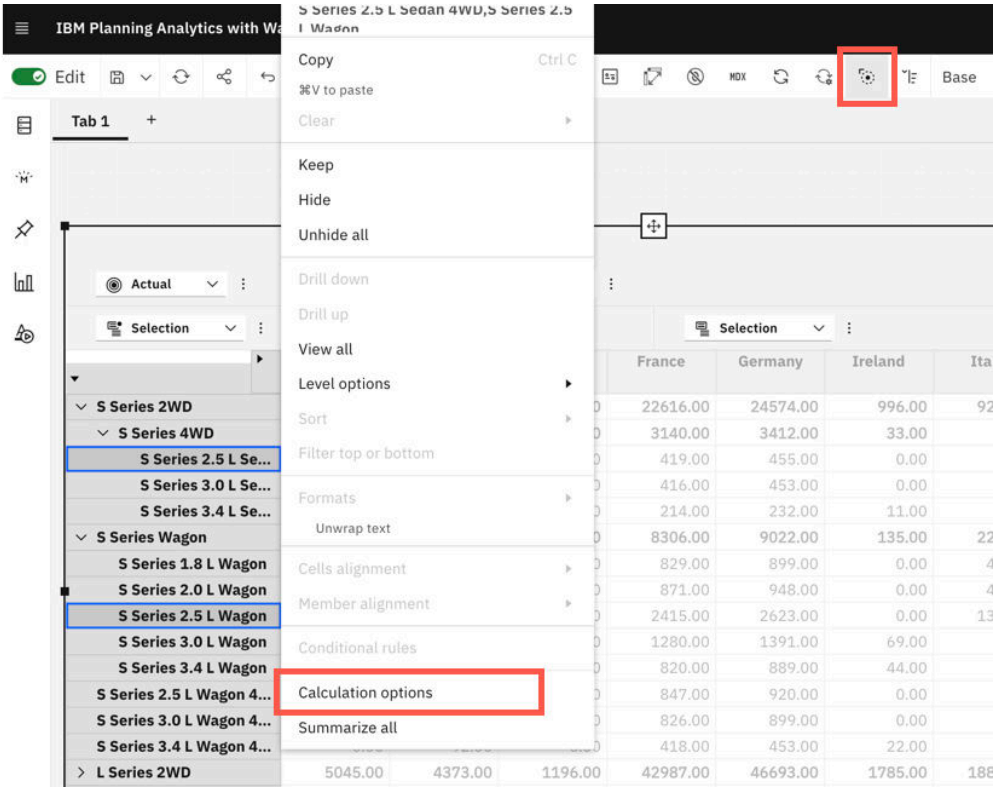
An asymmetric calculation can be applied within a single nested set or span multiple nested sets of the same dimension. Note, an asymmetric calculation is calculated by using the specific members that you select in the context of a nested dimension.

## Procedure

1. Click the **Target selection** icon in the toolbar or press **F7**.

You are now in target selection mode.

2. Select the members that you want to be included in the calculation. Use Ctrl+click to select multiple nonadjacent members, use SHIFT+click to select multiple adjacent members. The asymmetric calculation is inserted next to the last member you select.
3. Right-click a selected member, then click **Calculation options**.




4. Enter a descriptive name for the calculation.

   If you create a calculation without entering a descriptive name, the calculation is automatically assigned a name that reflects the calculation type you choose, with references to the members included in the calculation. For example, **Golf Accessories + Putters** or **median (Irons, Putters, Woods)**.
5. Select the calculation that you want to apply. See “Member calculations” on page 630 for more details on the calculations that are available.
6. Click **OK**.
7. To remove a calculation, right-click the calculation label in the view, then click **Remove**.

## Calculation formats

When a calculation is added to a view, TM1 formatting is applied to the calculation.

The formatting that is applied to a calculation is determined in the following order of priority.

1. The format of the selected members in the calculation. If the calculation uses a range selection, then the format is limited to the start and end member of the range.
2. The format of the overview context members, going from right to left.

   **Note:** In versions earlier than 2.0.48, the format of the overview context members is determined by going from left to right.
3. The format of members on the bench, going from bottom to top.

   **Note:** In versions earlier than 2.0.48, the format of members on the bench is determined by going from top to bottom.

4. If no format is found for a member, then a default format of `#,##0.00;(#,##0.00)` is applied. For example, the value 256694.89 is formatted as 256,694.89.

### Exceptions

Rank calculations always use the format `#,##0`.

Percentage calculations always use the format `##0.00%;-##0.00%`

## Rename a calculation

Though you can assign a name when you create a calculation, you can change the name of an existing calculation.

### Before you begin

You must be in the Exploration visualization type to rename a calculation.

### Procedure

1. Right-click the row or column label of the calculation you want to rename. On a mobile device, click the row or column label, then click  .
2. Click **Rename**.
3. Enter a new name for the calculation.
4. Click **OK**.

## Delete a calculation

You can delete one or more calculations from the rows or columns of a view.

### Before you begin

You must be in the Exploration visualization type to delete calculations.

### Procedure

1. Right-click the row or column label of the calculation you want to delete. On a mobile device, click the row or column label, then click  .

   If you are using a desktop browser, you can delete multiple calculations on a row or column simultaneously. CTRL-click or SHIFT-click the labels to select the calculations, then right-click one of the selected labels.
2. Click **Remove**.

# Find data

Find data in IBM Planning Analytics Workspace by using the intent bar or the data tree.

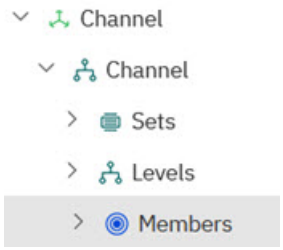
## Find data with the tree

The data tree shows all of the TM1 databases that are configured for Planning Analytics Workspace, and the available cubes, dimensions, hierarchies, sets, levels, views, and members. The data tree is hidden by default.

You can view the data tree while you work in a book. To display the data tree, click the **Database** icon on the vertical bar: . When the tree is visible, you can click the **Database** icon to hide the tree.

Click the **Settings** menu to manage the behavior of the data tree.

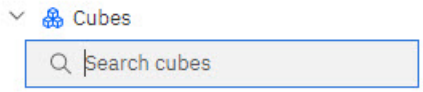
You can use the **Display members** option to determine whether the Members node appears in the data tree under dimension hierarchies. When the option is enabled, the Members node is available, as in the following example.




When the option is disabled, the Members node is not available in the data tree.

To locate an object within a long list of items in the data tree, you can search at the node level.

When you expand the Cubes, Dimensions, Sets, Processes, Chores, or Control Objects:Cubes nodes in the data tree, you can see a search box with the node name. You can also see a search box when you expand the Applications nodes if the folder contains at least 10 items. For example, the following search box appears when you expand the Cubes node:




Type in the box to see the objects that contain your search string. The results are updated continuously as you enter characters.

# Sets

Sets are a group of members within a dimension. You can create and edit sets through the **Set editor** to limit the number of members that you see in a view.

You can also create sets from hierarchies.

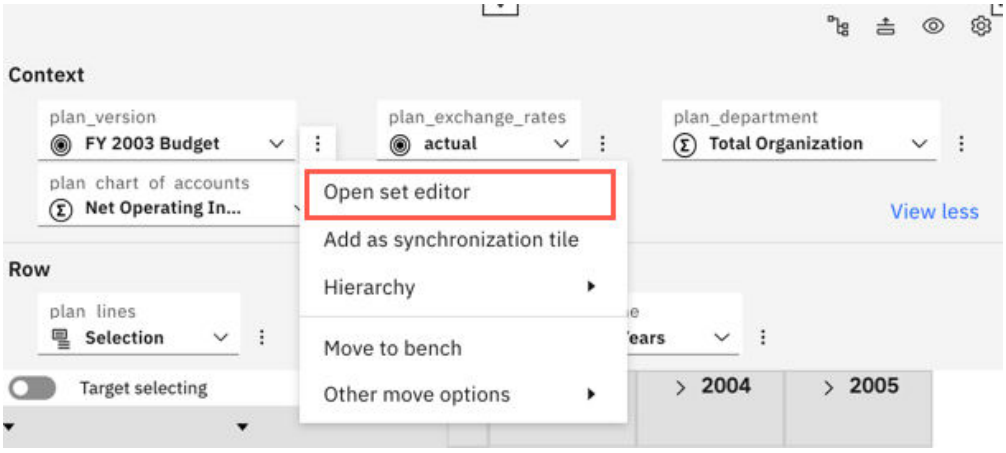
## Create and edit a set

A set is a collection of members in a dimension that limits the number of members that are displayed in a view.

You can create or modify an existing set by using the set editor.

To create a set, right-click a dimension or hierarchy in the content tree, and select **Create set**. The set editor opens. Now, follow steps 1 and 2 to create a set.

To edit an existing set, click the menu icon next to a dimension in the view and select **Open set editor**. You can also double click the dimension tiles or click the **Set Editor** icon in a dimension menu to open the set editor.




Once you open the set editor, follow these steps to create or modify a set:

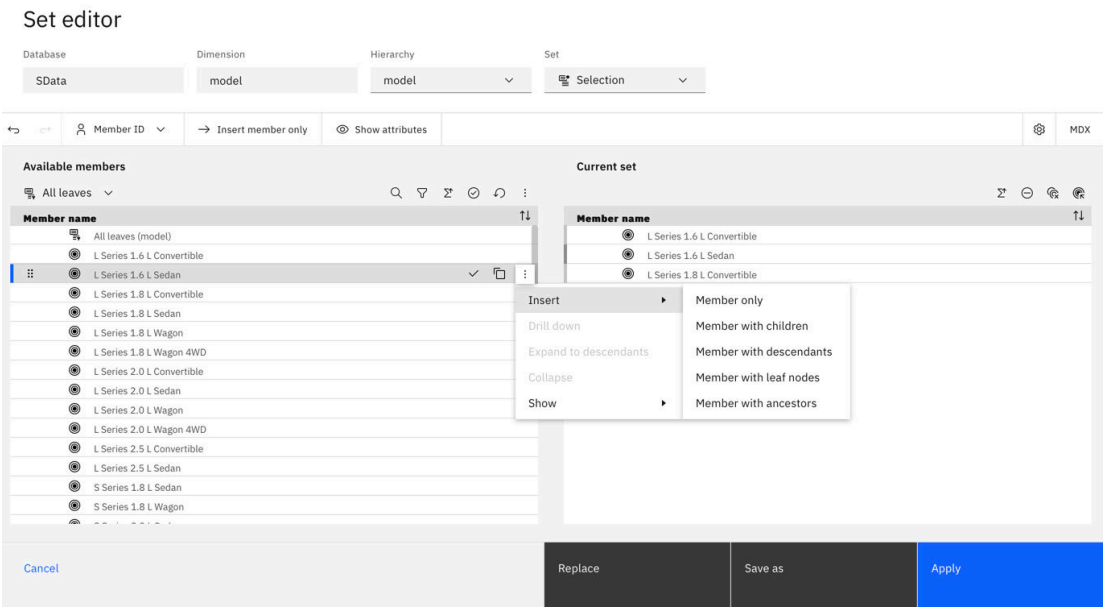
1. Display the members that you want to use in the set by selecting from the **Hierarchy** menu, and the **Set** menu.

   **Note:** You can also select a virtual set from the **Available members** menu to see **All members**, **All roots**, or **All leaves**.

2. Select the members in the **Available members** list, then choose one of the following options to add the members to the **Current set**:

   - Drag the members in the **Available members** list to the **Current set**.
   - Click **Insert** to insert the members into the current set.
   - Click **Replace set with new insertion** to replace all of the members in the current set.
   - To append the members in the **Available members** list to the members in the **Current set**, select the member and then click **Insert**.

     To control the members that are inserted into the set, click the menu for a member and select one of these options under **Insert**.



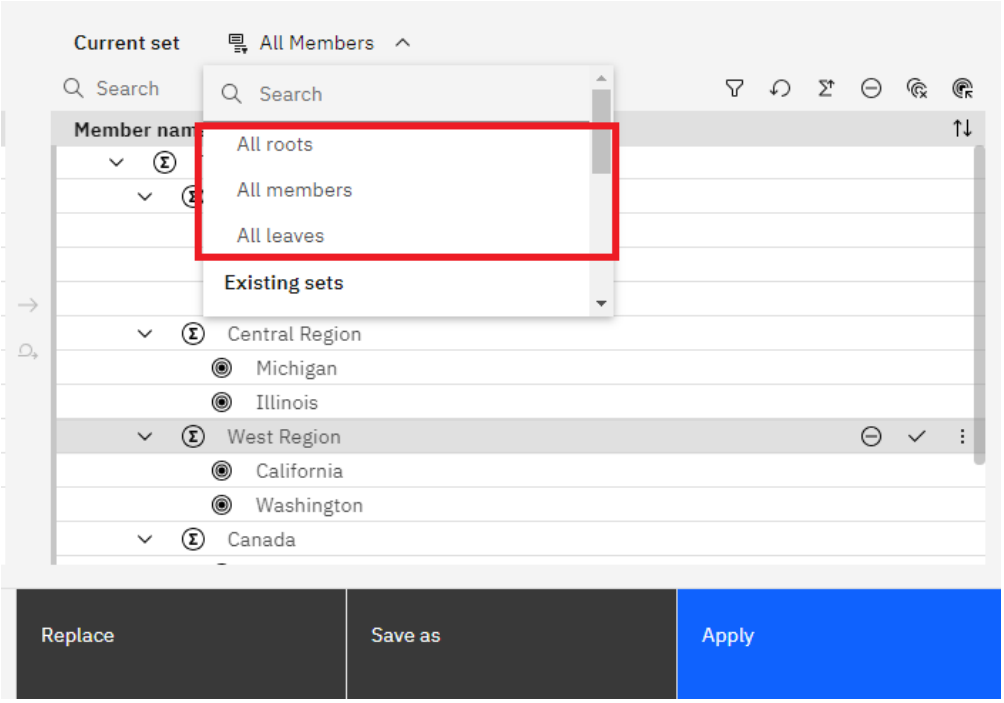

   - Click **Remove duplicated members** to display only distinct members from the current set.

## Working in the Current set

The **Current set** changes when you:

- Select a named set or virtual set from **Current set** list
- Replace **Current set** with a named set from the **Available members** pane
- Switch hierarchies - when you switch hierarchies, the **Current set** changes to the default named set. If a default named set does not exist, the **Current set** switches to **All roots**.
- Delete the current named set

You can select a named or virtual set from the **Current set** list. The availability of sets in the **Current set** pane lets you work with these sets without having to toggle **Available members** back on when you are in single-pane mode.




When you select a virtual set, the **Current set** label changes to display the virtual set's name. If your **Current set** is a named set, the **Current set** keeps the alias of the existing set and expands above with the virtual set, when you select **Show totals trailing** and choose a virtual set.

You can do the following to make changes in the **Current set**:

**Change the position of a member**

Right-click the member, then click one of the **Move** options. You can only change the position of a member if the set is static. See "Convert a dynamic set to static" on page 644.

You can also cut, copy, and paste members (use CTRL + V for Windows, or CMD + V for the Mac).

**Remove members**

Select the members, then click **Remove**.

To remove all members from the **Current set**, click **Remove all members**.

You can also remove duplicate members from the **Current set** by clicking **Remove duplicated members**.

**Keep only selected members**

Select the members, then click **Keep**.

**Display Member ID**

You can display the **Member ID** or the **Alias** for a member instead of the **Caption**. An alias is an alternative name for a member. Click **Member ID** and select the display name type.

**Sort members**

You can sort members in the **Current set** in the following way by clicking **Sort**:

- Ascending

- Descending
- Hierarchically ascending
- Hierarchically descending

**Undo and redo**

To undo an action in the **Current set**, click **Undo**. To redo that action, click **Redo**.

**Reset the current set**

You can use **Reset the current set** to reset changes that you made to the currently selected set.

You can also use the **Reset the current set** button in the set editor to refresh the set even if there are no changes to the set.

## Apply and save your changes

1. To save your changes as a new set that can be reused in other views, click **Save as**.
2. Give the new set a name and select **Public** to share the set with other users. Otherwise, select **Private**.
3. If you are editing an existing set, you can apply the changes to your view without saving the new set configuration. Click **Apply**.

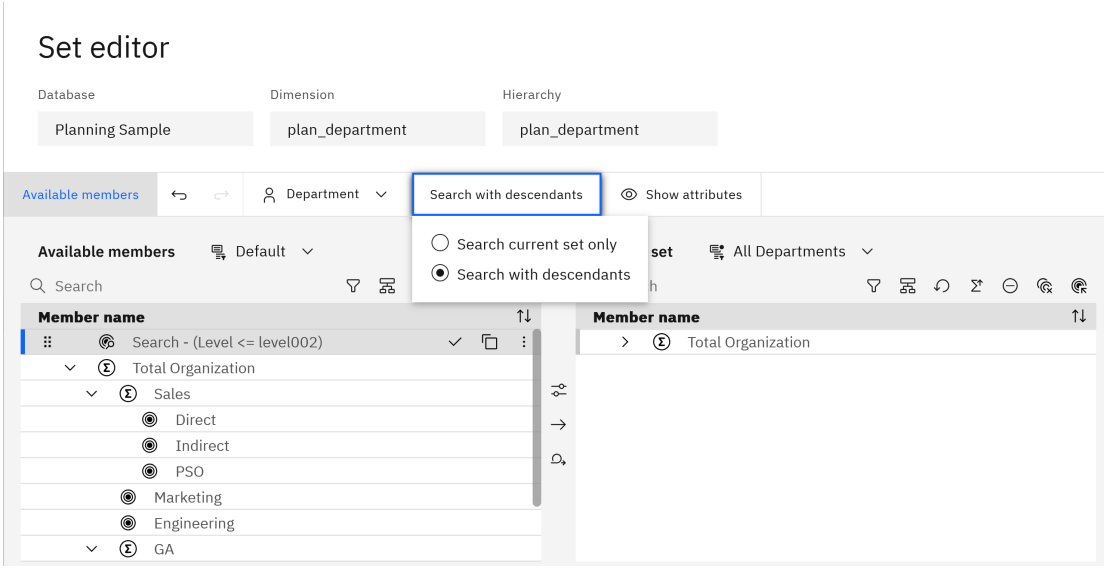
# Searching in sets

You can search in the set editor when the list of members is large or you are unsure of an exact member name. You can also search to include members in the set that match specific criteria.

## Searching in the Available members panel

To search for available members:

1. Open the set by double-clicking the dimension tile or click the menu next to the dimension and select **Open set editor**.
2. If you want to search only the current set members, select the **Search current set only** option. If you want to include the descendants of consolidated members in the search, select the **Search with descendants** option.




3. Type your search criteria in the search box.
4. Press Enter to search.

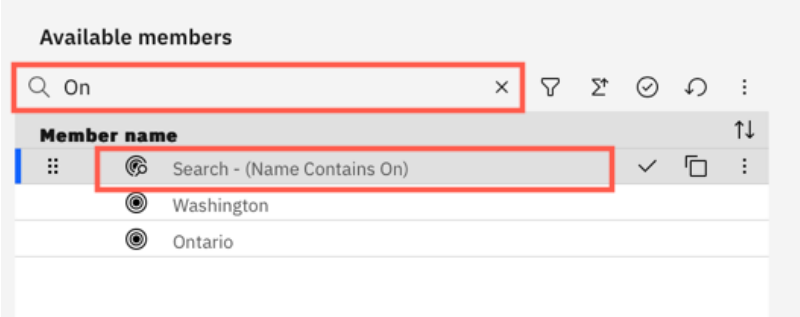
The **Available members** shows all members that match the search and a member that is named **Search** that indicates the search criteria.

5. Click **Insert** or **Replace set with new insertion** to add search results to the **Current set**.

Search looks for the search criteria in the current MDX only. This means that if a set is filtered, search looks for members that match the search criteria in the filtered set and not the full list of members.

**Note:** You can clear your search by deleting or replacing the input in the search box and pressing Enter. Or, click **Reset**.

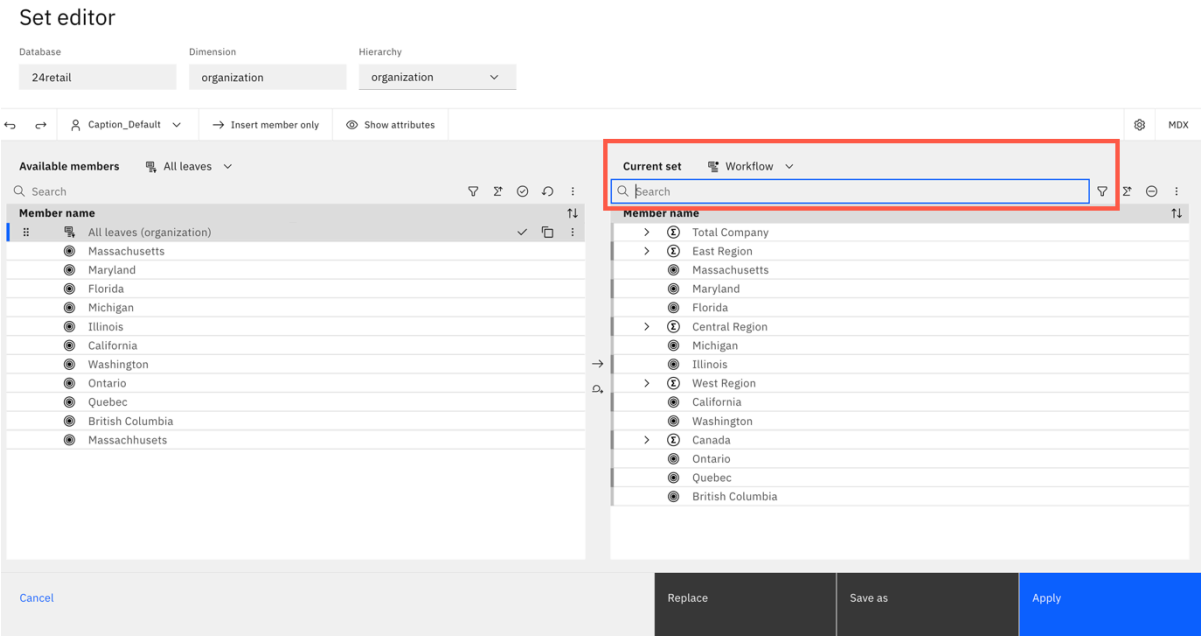
The following example shows a search for members with names that contain the characters on.




When you add the member **Search - (Name Contains on)** to your current set, you create a dynamic set that includes all the current members. Additionally, the set includes any future members that contain on in their name. A dynamic set contains a query that runs every time you open the set. If the parent dimension for the set contains a new member that matches the search, the current set includes the new member the next time you open the set.

## Searching in the Current set panel

You can also search for members in the **Current set**.




Use **Search** to look for members in the current MDX state for the **Current set**.

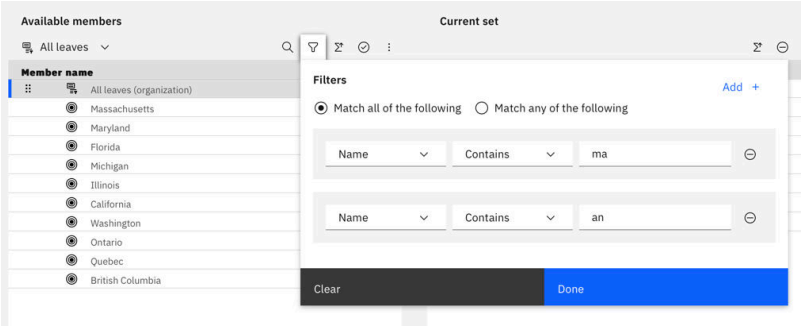
Click **Undo** to clear your search and return to the original current set.

If you want to search only the current set members, select the **Search current set only** option. If you want to include the descendants of consolidated members in the search, select the **Search with descendants** option.

## Filtering sets

You can filter a set by one or more attributes to choose the members that appear in the set. You can apply filters to the **Available members** and the **Current set** lists in the set.

You can filter on name, level, or attribute values and choose operators (such as =, <, >, <=, =>) to create dynamic selections. You can combine up to three filter parameters in one go.

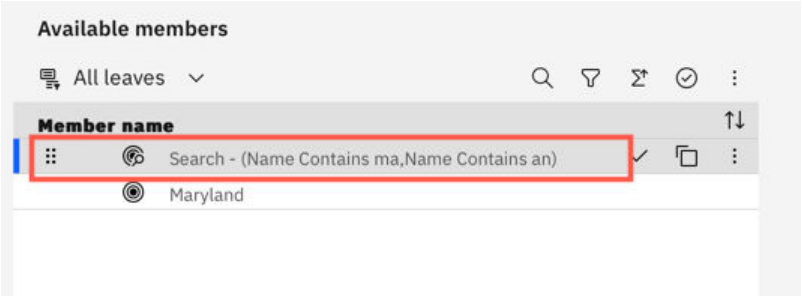



To filter a set:

1. Open the set by double-clicking the dimension tile or click the menu next to the dimension and select **Open set editor**.
2. Under **Available members** or **Current set**, click **Filter** and enter your filter parameters.
3. If you want to filter only the current set members, select the **Search current set only** option. If you want to include the descendants of consolidated members in the filter, select the **Search with descendants** option.
4. Select **Match all of the following** or **Match any of the following** to apply the filter statements to the set. The default selection is **Match all of the following**.
5. Click **Done**.

   The **Available members** or **Current set** now shows all members that match the filter statements.

In the following example, **Available members** was filtered to include only members that contain `ma` and `an` in their name.



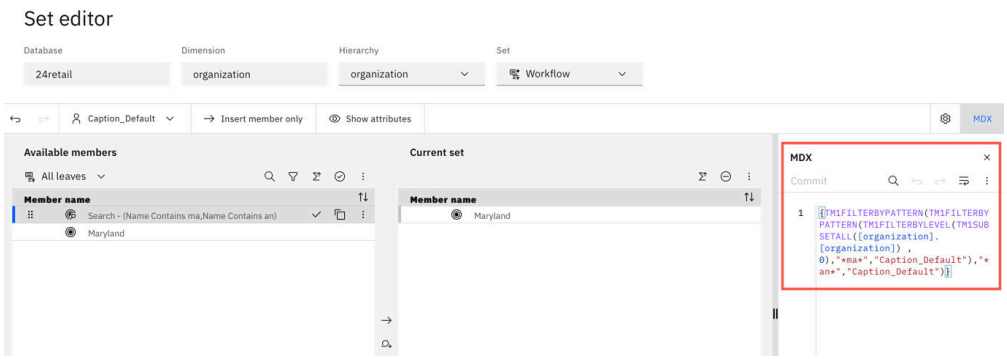

You can add the filtered results from **Available members** to the **Current set** by clicking **Insert** or **Replace set with new insertion**.

Once a filter is applied, the filter parameter clears from the **Filter** menu. This allows you to add more filters to further narrow your member selections.

For **Available members**, you can see the applied filters in the **Member name** list under or in the **MDX**.

**Note:** Replace the **Current set** with the filtered set and click **MDX** to see the applied filters.

Click **MDX** to see filters applied to the **Current set**.




When you apply a filter to an already filtered set, the new filter applies on top of the first filter.

For example, in the following MDX, a new filter was applied to an already filtered set to include only members with names that contain the letter `r`.




You can click **Reset** or **Undo** to clear all the filters you applied and restore the set to its original state.




## Narrowing down selections in the set editor

If you have large dimensions, you can make it easier to find what you are looking for in the set editor by narrowing the selection in the **Available members** panel.

You can filter selections in the following ways:

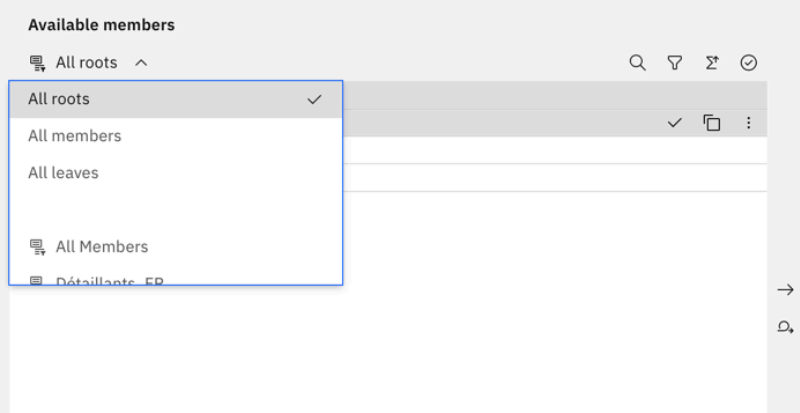
**Restrict the content in the set editor to the selected members and their descendants**

Select the members in the **Available members** panel and click **Keep**.

A selection is created that replaces **All members**. You can search in this selection, and you can use the selection to create a set.

**Select one of the default sets**

You can choose to display **Default set**, **All roots**, **All members**, or **All leaves** by default.



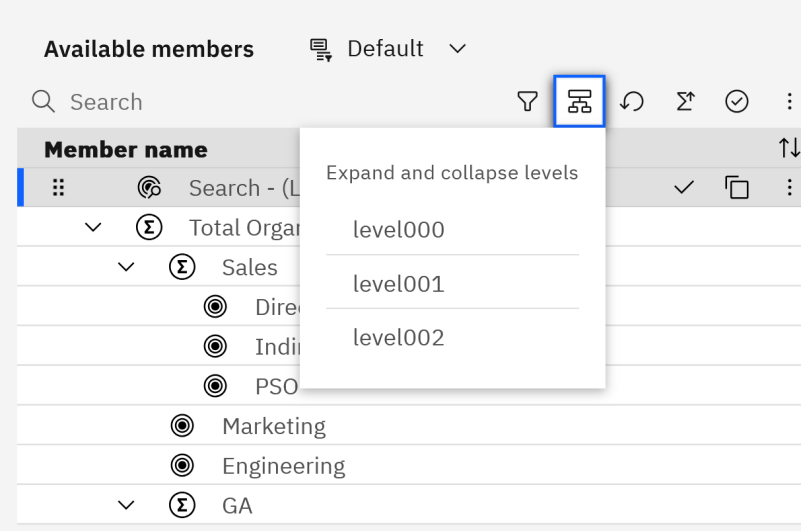

**All roots**
Contains the root members of the hierarchy. The leaf members are hidden but can be displayed by clicking. When **All roots** is searched, all members, including the leaf members, are searched.

**All leaves**
Contains the leaf members of the hierarchy. When this set is searched, only the leaf members are searched.

**All members**
Contains all members in the hierarchy, both consolidation and leaf members.

**Expand or collapse all consolidations at once**

Use the **Expand and collapse levels** button to expand or collapse consolidations up to the specified level. For example, you can collapse all members and then expand only a specific consolidation to examine the consolidation path.




From the **Menu** of a selected member, you can also:

**Drill down on one or more members**
Select **Drill down** from the menu to drill down on the selected member. This option creates a selection that can be searched in or used to create a set.

You can also double-click a member to drill down.

**Expand selected members and show their descendants**
Select **Expand to descendants** to show the all the descendants for the selected member. To collapse one or more expanded members, select the members, and from the **Menu**, click **Collapse**.

**Show all ancestors**
Select **Show** and choose **Ancestors** to show the selected member and its ancestors. Other members are hidden. This option creates a selection.

**Show all decendents**
Select **Show** and choose **Descendants** to show just the selected member and its descendants. Other members are hidden. This option creates a selection.

**Show immediate parents**
Select **Show** and choose **Parents** to show just the selected member and its immediate parents. Other members are hidden. This option creates a selection.

**Show all children**
Select **Show** and choose **Children** to show the selected member and its children. Other members are hidden. This option creates a selection.

**Show all leaves**
Select **Show** and choose **Leaves** to show the selected member and all its leaves. Other members are hidden. This option creates a selection.

## View properties and attributes in a set

You can view any defined properties and attributes for members in the set editor.

For example, you can view properties such as:

**Weight**
The weight property controls how a child member rolls up to its immediate parent, whether that child is also a parent of another consolidation or a leaf member. Top level consolidations do not have a weighting.

**Level**
Show the level of the member in the hierarchy.

**Type**
See whether a member is consolidated or simple (leaf).

**Index**
The display order of members in a dimension.

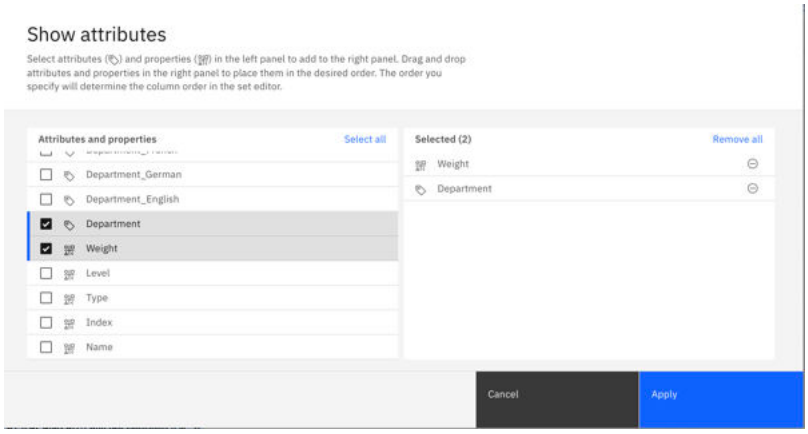
**Name**
The member ID.

**Note:** Attributes in the set editor can help you select members when creating a set but they do not automatically display in the view. To display attributes in a view, see Show member attributes in the cube view.

## Procedure

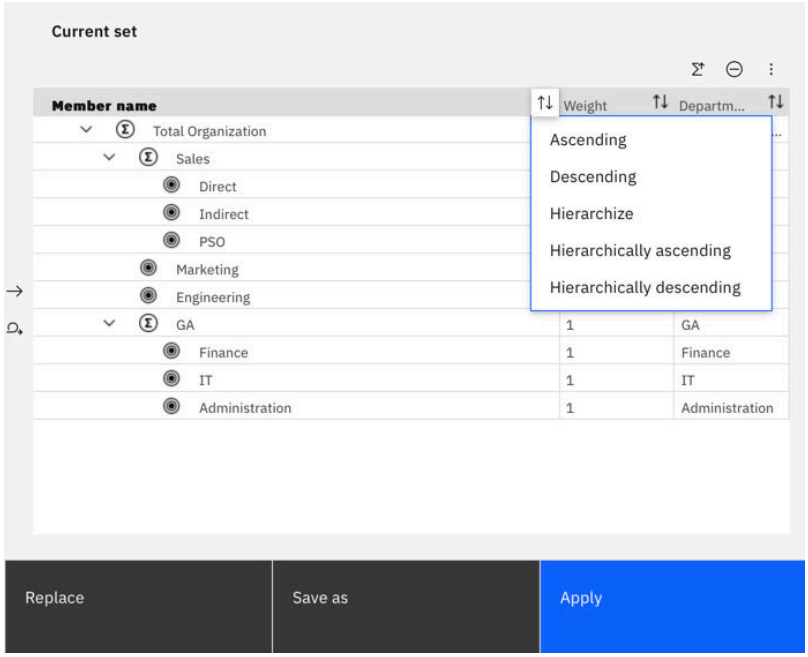
1. In the set editor, click **Show attributes**.
2. Select attributes and properties in the **Attributes and properties** panel to add them to the **Selected** panel.

   The order in which you select attributes and properties determines the column order in the set editor. Scroll over the selected attributes and properties and use the arrows to change the order.



3. Click **Apply**.

   In the **Current set**, you can sort attributes and properties with the **Sort** icon. The **Hierarchize** sort option reverts any sorting you applied to the members.




## Convert a dynamic set to static

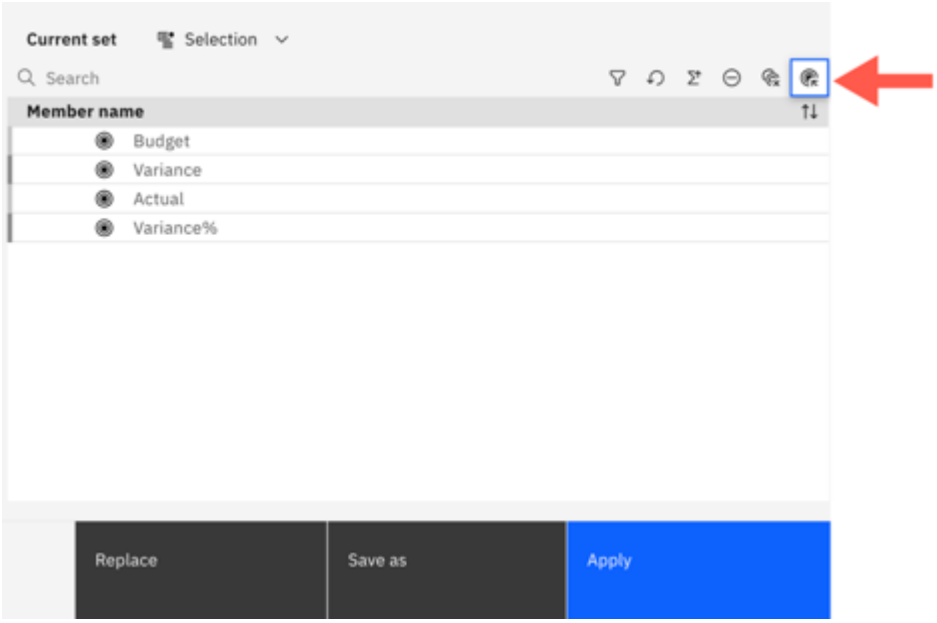
You can convert a dynamic set to a static set in the set editor. When you convert to a static set, the MDX expression that generated the dynamic set is deleted and the set contains only the members that are present at the time of conversion.

### About this task

Converting a dynamic set to a static set is useful when you added a calculation to a dynamic set, but want to control the positioning of the calculation in the set. When you add a calculation to a dynamic set, the position of the calculation in the set is determined by the MDX expression and the position cannot be changed. When you convert to a static set, you can move the calculation to any location in the set.

### Procedure

1. Open the dynamic set in the set editor.
2. In the **Current set** pane, click **Convert to static**.




3. Click **Apply**.

## Show dialog notifications in the set editor

The **Show dialog notifications** option in the set editor **Settings** controls the display of certain prompts within the set editor.

When the **Show dialog notifications** setting is enabled, you will see the following prompts when you:

- Drag a member from **Available members** into a dynamic set in the **Current set**
- Re-order members in a dynamic set
- Convert dynamic sets into static sets
- **Reset** the MDX script after making changes to the MDX

You can turn off display for one or both of these prompts by selecting the **Don't show this message again** option within the prompts.

If you turn off display for one prompt, the other prompt still displays when you perform the related action. In that case, the **Show dialog notifications** setting is in a partially enabled state.




When you select the **Don't show this message** option for both prompts, the **Show dialog notifications** setting is disabled completely and both prompts no longer display.




## Show distinct ancestors in the set editor

The **Distinct ancestors** option in the set editor **Settings** controls the display of parent and ancestor members when you expand selections.

Use the **Distinct ancestors** option in the set editor **Settings** to specify how parent and ancestor members are displayed when you use the **Expand to** or **Keep with** options.

If you clear the **Distinct ancestors** setting, multiple instances of the same ancestor might appear if a selected ancestor has multiple parents.

If you select the **Distinct ancestors** setting, only one instance of each ancestor appears. This setting works by automatically applying the **`MDX DISTINCT`** function to the query.

## Deleting sets

You can delete sets that are not currently used in any views.

If you are a TM1 administrator, you can delete any public or private set that you own. If you are not an administrator, you can delete only the private sets that you own.

You can delete sets from the following places in a book:

- Navigation tree
- Set editor

**Important:** Deleted sets cannot be restored. After you a delete a set, the set disappears from both the navigation tree and the set editor.

### Deleting a set from the navigation tree

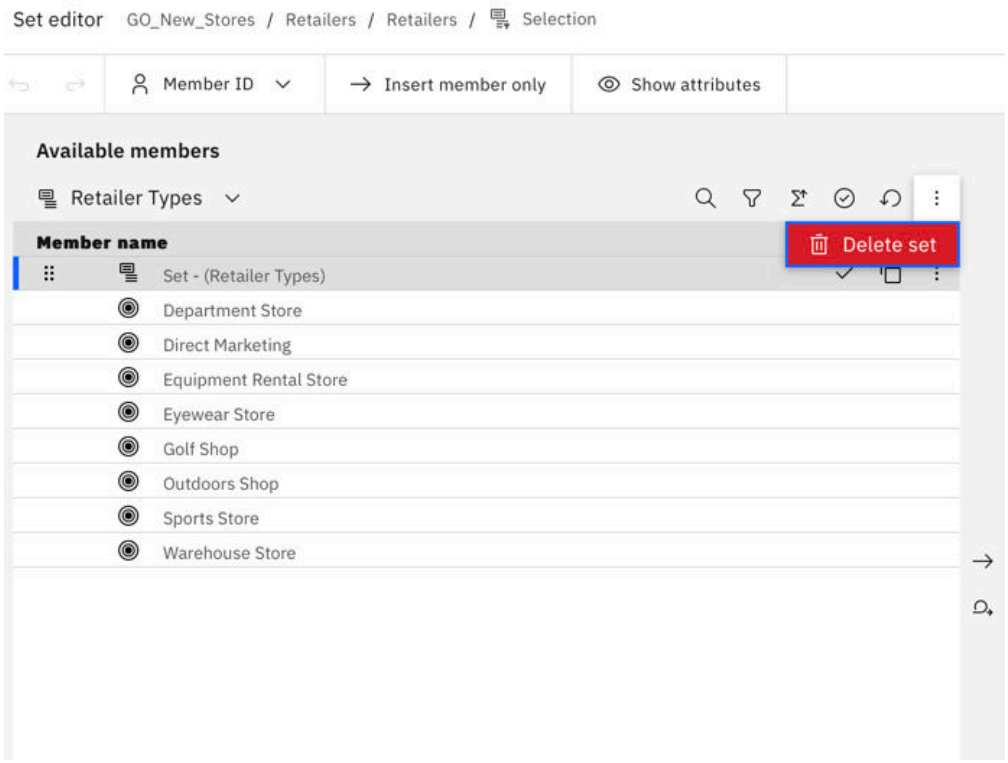
1. In the navigation tree, locate the set you want to delete.

   The path to a set is ***<Database_Name>*** > **Dimensions** > ***<Dimension_Name>*** > ***<Hierarchy_Name>*** > **Sets** > ***<Set_Name>***.
2. Right-click the set, then click **Delete set**.

### Deleting a set from the set editor

1. From a new experience view, open the set editor.
2. Locate the set you want to delete.

   **Note:** Unnamed sets such as **All roots**, **All members**, and **All leaves** cannot be deleted.
3. In the toolbar on the **Available members** panel, click **More**.



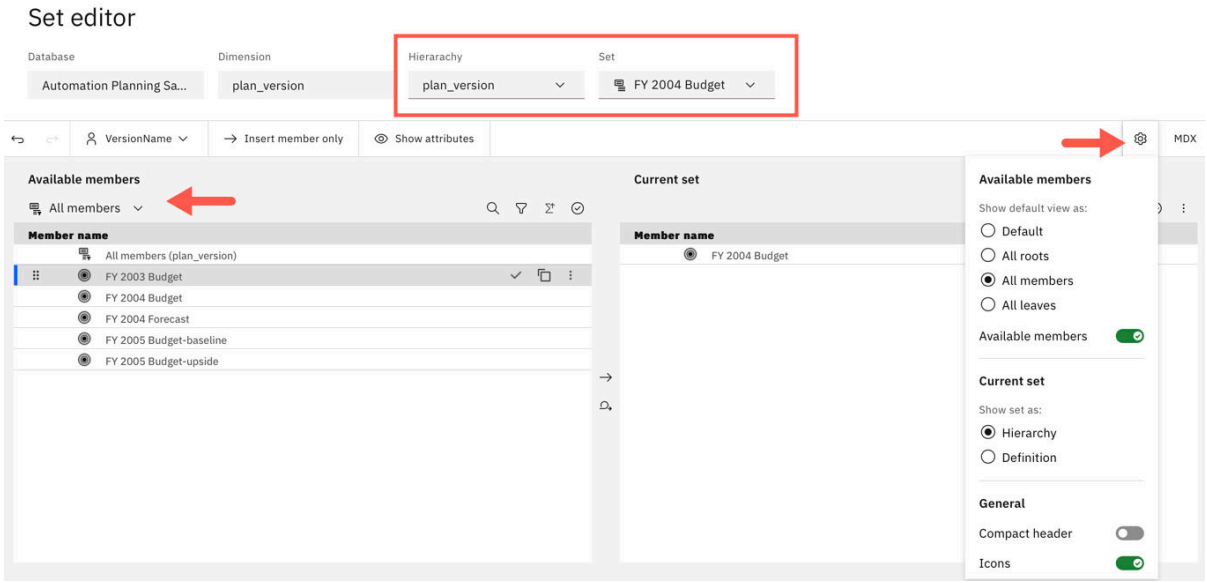

4. Click **Delete set**.
5. Click **OK** to confirm you want to delete the set.

# Editing a set (quick analysis)

You can edit a set in IBM Planning Analytics Workspace without saving for quick analysis.

## Procedure

1. Open the set by double-clicking the dimension tile in the view, or click the menu next to the dimension and select **Open set editor**.
2. Display the members that you want to create the set from by selecting from the **Hierarchy** menu. You can also change the set from the **Set** menu.
3. If your changes are simple, select the members that you want to be in the set in the **Available members** menu. You can also access **Available members** from the settings menu.




4. Once your changes are done, click **Replace**. The set editor closes and you return to the view, displaying the members that you selected.

   **Note:** The **Replace** feature is available only if you have members selected in either the **Available members** or **Current set** pane.

   If you have members selected in the **Current set** pane, clicking **Replace** replaces members in the view.

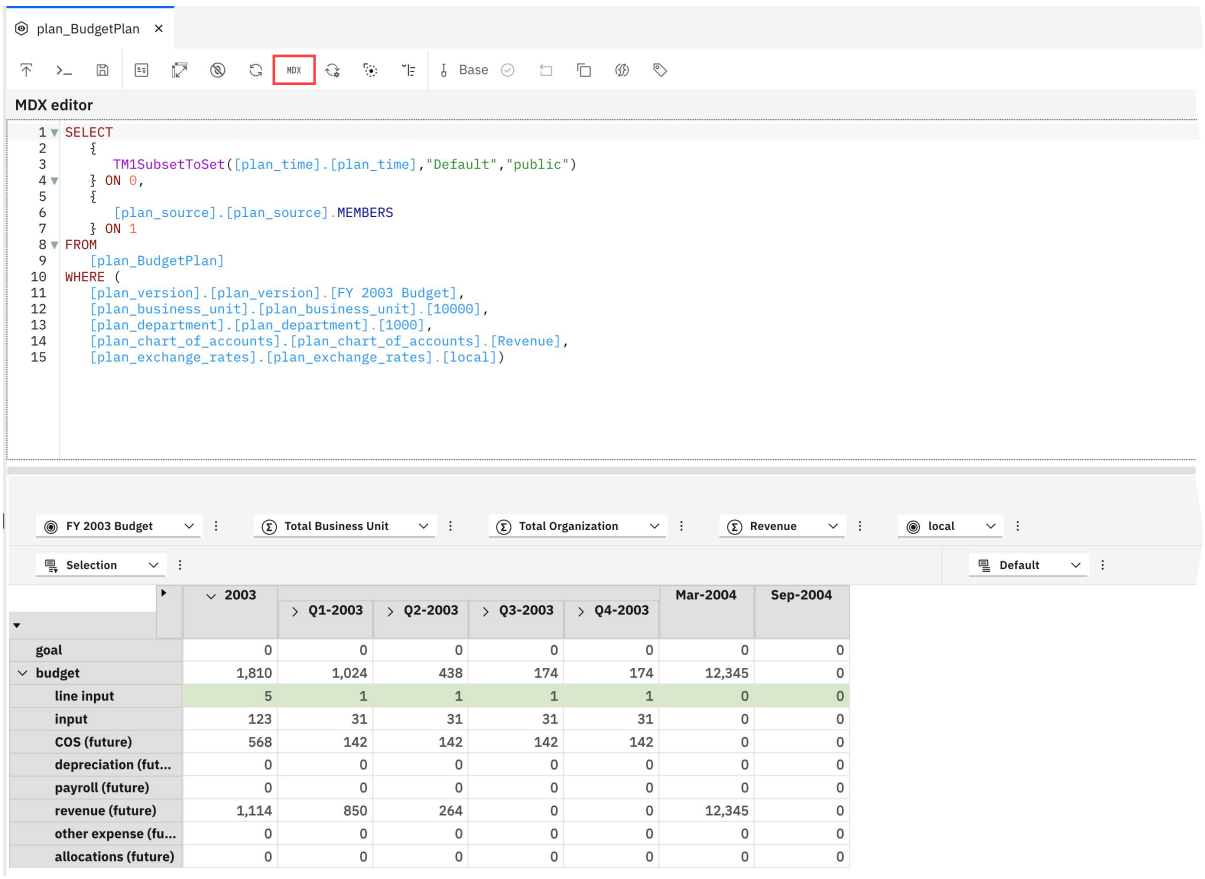
# Edit sets using the MDX editor

The MDX editor is available in the IBM Planning Analytics Workspace books and modelling workbench.

**Note:** The new MDX editor is not currently available in IBM Planning Analytics for Microsoft Excel.

The MDX editor allows you to make changes to the MXD expressions in a view.

To access the MDX editor, click the **Edit MDX** icon MDX.




| | 2003 | Q1-2003 | Q2-2003 | Q3-2003 | Q4-2003 | Mar-2004 | Sep-2004 |
|---|---|---|---|---|---|---|---|
| goal | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| budget | 1,810 | 1,024 | 438 | 174 | 174 | 12,345 | 0 |
| line input | 5 | 1 | 1 | 1 | 1 | 0 | 0 |
| input | 123 | 31 | 31 | 31 | 31 | 0 | 0 |
| COS (future) | 568 | 142 | 142 | 142 | 142 | 0 | 0 |
| depreciation (fut... | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| payroll (future) | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| revenue (future) | 1,114 | 850 | 264 | 0 | 0 | 12,345 | 0 |
| other expense (fu... | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| allocations (future) | 0 | 0 | 0 | 0 | 0 | 0 | 0 |

The following features are available in the MDX editor:

- Autocomplete: provides suggestions on dimension names, functions, and other TM1 objects while you are typing.
- Syntax coloring: shows different portions of the MDX statement in different colors for increased readability. For example, keywords are written in red, and tuples are written in blue.
- Undo: allows you to undo the last change.
- Redo: allows you to redo the last change.
- Find and replace: clicking ctrl+f open a dialogs that allows you to search for specific text in the MDX, and replace it.
- Indenting: the MDX editor now automatically formats the MDX script, indenting each line. When you make changes in the view, the MDX editor updates with the correct formatting. You can manually indent the lines in the script by clicking **Indent script** in the MDX editor toolbar.
- **Commit** applies your MDX edits to the view. When you click **Commit**, a text box appears telling you if the changes were made successfully.
- **Synchronize** replaces the MDX in the editor with the one represented in the view. After you click **Synchronize**, click**OK** to synchronize.

  **Note:** When you make edits in a view, the MDX editor automatically synchronizes to match these changes. However, when you make edits in the MDX editor, the view does not automatically synchronize to match these changes. You have to manually select where you want your edits to apply using the **Commit** and **Synchronize** buttons in the MDX editor toolbar.

# Add a set consolidation to a view

You can create a set consolidation that is based on named and unnamed set members.

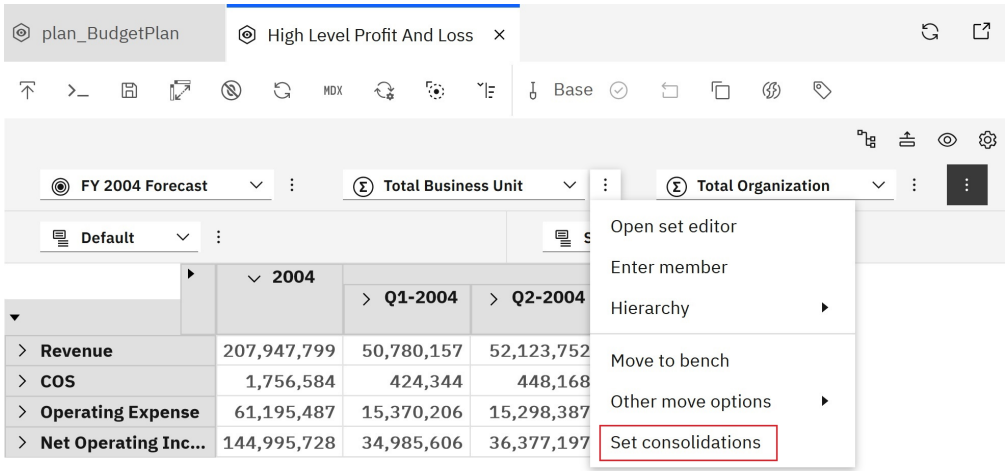
## About this task

A set consolidation can be created only in an Exploration visualization.

## Procedure

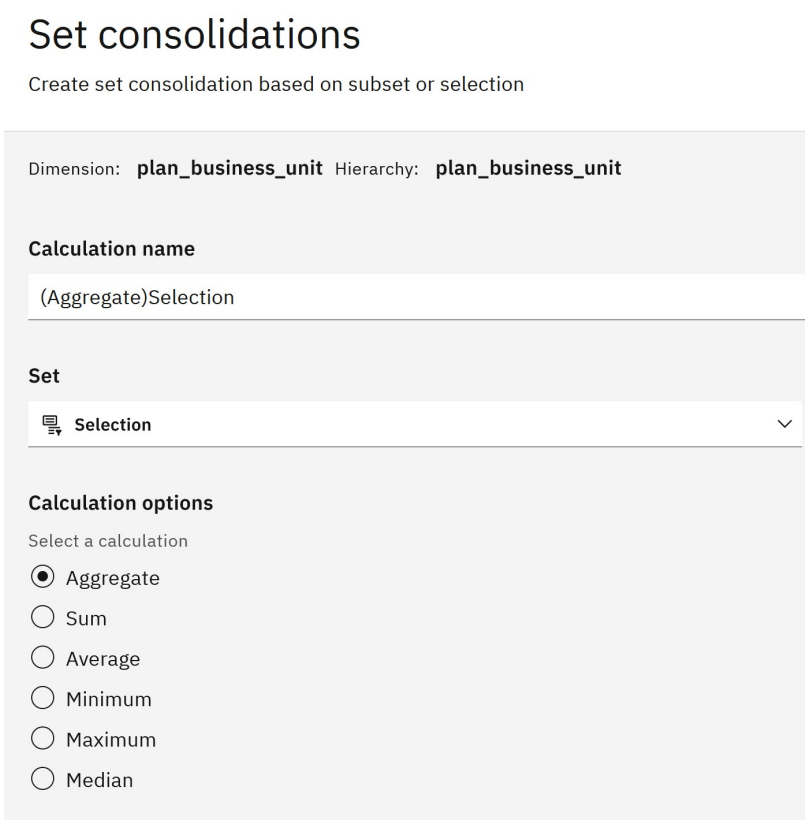
1. Locate the **Set consolidations** menu option in the context for which you want to create the set consolidation.

   This option is available from multiple places in the application: from the dimension tile overflow menu ⋮ (shown in the image below), from the column and row right-click menu, from the thumbs right-click menu, and also from the **Fields** panel tile menus.




2. Click **Set consolidations**.

   The following dialog box is displayed:

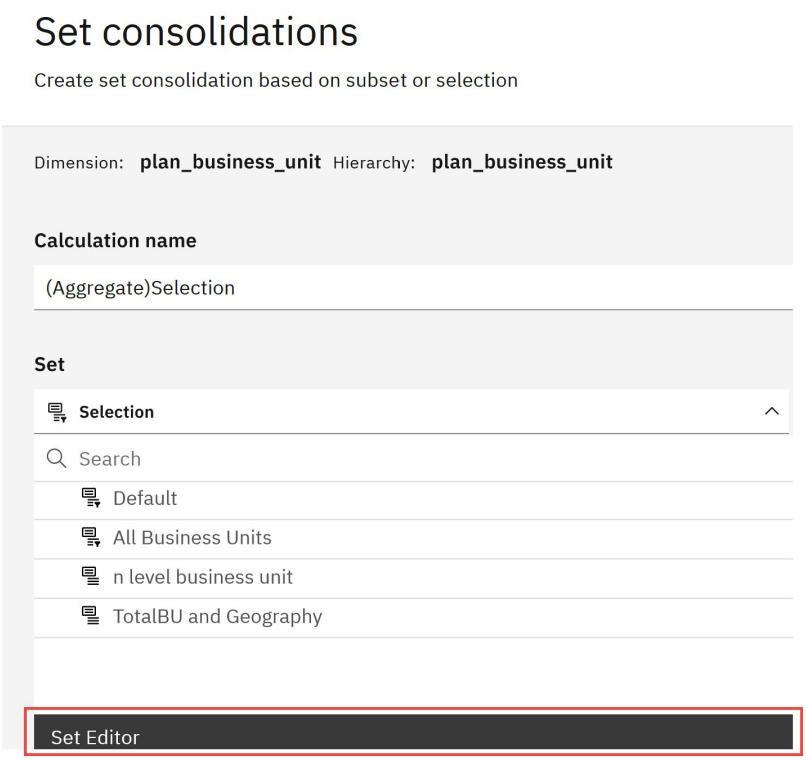



3. The **Calculation name** is populated based on your **Set** and **Calculation options** choices.

   If you type a custom calculation name, it will remain unchanged, no longer reacting to your selections.
4. In the **Set** box, you can select two types of sets:
   - Named sets

     These predefined sets are already listed in this box.
   - Unnamed sets

     Click **Set Editor** to select different members to add to the list of sets for your consolidation.




In the **Set editor**, select the members that you want to add.

**Tip:** Click **Apply** to create an unnamed set with the currently displayed member, or click **Replace** to create an unnamed set with the currently selected members.

For more information, see "Create and edit a set" on page 635.

The selected members are added at the end of the list in the **Set** box, and you can use them to create the consolidation.

5. Under **Calculation options**, select the calculation type that you want to create.

**Average**
Displays the average value of the set members.

**Minimum**
Displays the smallest value of the set members.

**Maximum**
Displays the largest value of the set members.

**Median**
Displays the median value of the set members (the middle value). If there are an even number of values, it displays the average of the two middle values.

**Sum**
Displays the total of the set members.

**Aggregate**
An aggregate calculation allows you to summarize values that can't simply be added up or averaged. For example, percentage or ratio values that are the result of a rule calculation.

Aggregate displays the aggregate value of the set members. This value is affected by whether a member has a C: level rule (a rule that applies to consolidated cells) associated with it or not. If the member has no C: level rule associated with it, Aggregate displays the sum. If the member has a C: level rule, the resulting value is the same as the consolidated member aggregating the same members. To use this feature, you must have IBM Planning Analytics version 2.0.9 or later.

6. In the **Set consolidations** dialog box, click **Apply**.

### Results

If you created the set consolidation for specific rows or columns, the consolidation is added in the grid after the related row or column. If you created the set consolidation from the rows or columns handle, the consolidation is added as the last row or column.

## Action Buttons

You can create an Action Button in IBM Planning Analytics Workspace that lets you execute a TurboIntegrator process and/or open a new destination with a single click.

You can configure a button to:

- Execute a TurboIntegrator process
- Open a sheet in the current book
- Open a sheet in a different book
- Open a URL
- Run a watsonx AI job

When you configure an Action Button to execute a TurboIntegrator process, you can also set the button to open a new destination after the process is complete.

**Note:** There are some differences between the behavior of Action Buttons in Planning Analytics Workspace and buttons in Planning Analytics for Microsoft Excel/TM1 Websheets:

- In Planning Analytics Workspace, the execution of a TurboIntegrator process from a button does not lock the tab or book. You can continue to work in the book or tab while the process is running. In contrast, executing a process from a button in Planning Analytics for Microsoft Excel or a TM1 Websheet locks the sheet; you cannot perform any actions until the process finishes running.
- You can't use Excel cell references to supply process parameter values in Planning Analytics Workspace.

### Procedure

1. To add an Action Button to a sheet, you must be in edit mode.
2. Click the **Action Button** icon  .

   If your sheet does not use a template (free form), a new button is placed in the first open area of the sheet. If your sheet uses a template, the button is placed in the first open defined region of the template. You can drag the button to other locations of your sheet.
3. Use the handles on the perimeter of the Action Button to resize the button as required.
4. Click the **Properties** icon, then click a category to set the display or action properties for the button.
   - **Text details** - Sets the style, font, alignment, and color of the button text.
   - **General** - Sets the fill color, border color, opacity, position, and size of the button.
   - **Configure** - Sets the action you want to execute from the Action Button. You can run a process, run an AI job, and navigate to a destination with the Action Button. For more information, refer to Configuring the Action Button to run an action.
5. Click **Save** to save the book with the new Action Button.

### Results

You can now click the Action Button to execute the action you defined.

## Configuring the Action Button to run an action

You can run a process, an AI job, or both from an Action Button, and navigate immediately or navigate after the run completes.

The configuration settings for Action Buttons are available under **Properties**. Enable any process to see the available settings.

You can configure each process and navigation to run on its own or in any combination with each other. For example, you can configure:

- A process to run and then navigate
- An AI Job to run after a process completes successfully
- An AI Job to run and navigate immediately
- A process to run and after successful completion, run an AI Job and navigation at the same time
- A process and an AI Job to run and then navigate after their successful completion.

### Run process

Use **Run process** to define the process that is run when you click the Action Button.

1. In **Edit** mode, select the Action Button that you want to configure and under **Properties > Configure**, enable **Run process**.
2. Select the database on which the process resides.
3. Select the process that you want to run.
4. If the process includes parameters that must be set before execution, click **Set parameters** to set the parameter values or configure user prompts that can be answered to set parameter values.
5. Select **Refresh after execution** if you want to refresh the current book after the process finishes running. The refresh updates any views, websheets, visualizations, or selectors in the target book.
6. If you want the process to use a file when it is run, select **Allow file upload**. This option allows you to upload a file to the database when you click the Action Button.

   **Note:** The process runs only if the file is uploaded successfully.
7. You can configure custom notifications to display the status of a TurboIntegrator process when it is run from an action button in a Workspace book.

   To customize notifications:

   a. In **Edit** mode, select the action button that you want to configure.
   b. Under **Properties > Configure**, go to the **Customize notifications** section.
   c. Choose the notification type that you want to customize:
      - **Success notification**: When the process completes successfully.
      - **Quit notification**: When the process exits early with a quit status.
      - **Warning notification**: When the process completes with a warning.
      - **Error notification**: When the process completes with an error.
      - **Running notification**: When the process is running.
   d. Enter the notification message for each status.
   e. Choose how long you want to keep the message visible in the **Time on screen** section.
8. Select one of the following **Execute second click options** to configure the action button for additional clicks:

   a. **Cancel process -** When an action button that is already running a process is clicked again, the process is canceled, and a related message is displayed. This is the default behavior, which is known from previous versions of the product.

b. **Do nothing** - When an action button that is already running a process is clicked again the process continues to run. A new process is not started.

c. **Run processes in parallel** - When an action button that is already running a process is clicked again a new process is started, and the previous process or processes continue to run. This option is useful when the book user wants to run a series of overlapping processes by using the same action button.

**Note:** You can run a maximum of 5 processes in parallel. If you attempt to run the 6th process, a message about this limit is displayed.

## Run an AI job

Use **Run AI Job** to define the job that is run when you click the Action Button.

1. In Edit mode, select the Action Button that you want to configure and under **Properties > Configure**, enable **Run AI Job**.
2. Select the deployment and job that you want to run.
3. Select **Refresh after execution** if you want to refresh the current book after the process finishes running.

## Navigation

Use **Navigation** to specify the destination that is opened when you click the Action Button.

1. In Edit mode, select the Action Button that you want to configure and under **Properties > Configure**, enable **Navigation**.
   a. To navigate immediately upon clicking the Action Button, choose **Immediately** and select the destination.
      i) To go to a tab in the current book, click **Tab**, then select the tab that you want to open from the list of available tabs.
      ii) To go to a tab in a different book, click **Book** and select a book, and the tab you want to open. Click **Pass context** to pass context for any items that have synchronization that is enabled in both the current tab and the destination tab.

      If you configure an Action Button to navigate to a tab in another book, you can rename the target book and/or sheet without having to update the button properties. The button automatically resolves any name changes to target tabs or books.
      iii) Click **Refresh target book** option to refresh a target book you are navigating to, if the book is already open. When the **Pass context** is also enabled, context passes for any items that have synchronization enabled and the target book refreshes. If **Pass context** is enabled but no context is being passed on navigation, the target book still refreshes, if it is already open.
      iv) To open a URL, click **Hyperlink**, then enter a fully qualified URL.
   b. To navigate after a process completes, select **After process completes** and select where you want to navigate to when the process completes successfully and when the process completes with errors.

   If you run a process and an AI job, then both must complete successfully for you to select the navigation options for when the process completes successfully. If the process or the AI job fails, then you need to specify the navigation options for when the process completes with error.

# Set process parameters for a button

If you configure a button to run a TurboIntegrator process that requires input parameters, you must configure questions that can be answered to provide parameter values or specify default parameter values.

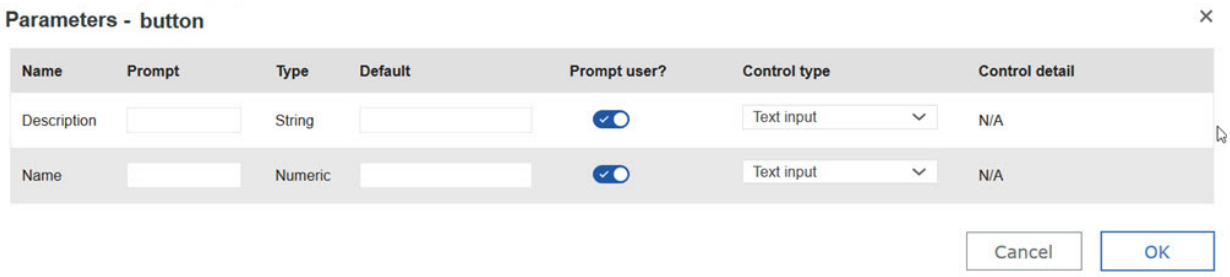
## About this task

The names and types of parameters for a TurboIntegrator process are included in the process definition; these parameters are not defined by the button properties. However, when the process is run from a button, values for the parameters must be passed into the process. You can configure a button to pass default parameter values into the process without requiring user input or you can configure prompts that ask the user to provide parameter values.

## Procedure

1. To set process parameters for a button, you must be in edit mode.
2. After adding the button to a sheet, click the button to select it. You should see several sizing handles on the perimeter of the button indicating that it is selected.
3. Click the **Properties** icon , enable **Run process**, then select a database and a process.
4. Click **Set parameters**.

   If the process does not include any parameters, you see a message that says `No data available in table`. In this case, you can click **Cancel** - you don't need to set any parameter values before running the process.
5. Enter a parameter title and add a description for a process that requires parameter values and go to the **Set parameters** tab.




   The name and type for each parameter is defined in the process. You can't change these parameter properties, but knowing them helps you create a relevant prompt and set default values.
6. For each parameter:
   a) Enter a **Prompt** that the user will respond to when providing a parameter value. This should be a simple statement or question that clearly indicates what kind of response is required from the user.
   b) Set the **Prompt user** option. Select this option to display the prompt that you created when the process is run from the button. The value that the user provides in response to the prompt is passed into the process. Deselect this option to pass the default parameter value directly into the process. Default parameter values are describe a bit later in this topic.
   c) Select a **Control type** and if necessary, set **Control detail**. These options determine how the user responds to the prompt to set the parameter value. The Control types available vary according to the parameter type.

      For string parameters, the following Control types are available:

      **Text input**
      Presents a box where the user can enter the parameter value. You do not need to specify any **Control detail**.

**List**
Presents a list of values from which the user can choose the parameter value. You must enter the values that you want to appear in the list in the **Control detail** column, with all values separated by commas.

**Member list**
Presents a list of dimension set members from which the user can choose the parameter value.

**Note:** The **Member list** parameter was previously called **Dimension list**.

To define the **Member list** parameters, click the **Edit** icon. Pick the dimension that contains the set you want to appear in the list and the hierarchy that contains the set you want to use.

You can also choose a subset that contains the members that you want to appear in the list. If you don't select a subset, then it is the member name that passes and the user clicking the Action Button will select from all of the members without aliases.

You can pass either the member name or member unique name in the TurboIntegrator process. The member name passes by default.

**Set list**
The **Set list** passes a subset to the TurboIntegrator process.

To define the **Set list**, click the **Edit** icon and pick the dimension and hierarchy. The user can then choose from the list of subsets in the hierarchy when they click the Action Button to run the process.

You can choose to show the set editor in the parameters prompt which allows the user to open the set editor and generate a selection instead of using one of the named subsets.

**Synchronized**

The process automatically picks up parameter values from other synchronized items in a book.

Select the dimension and hierarchy to be synchronized under the **Control Details** column.

A process can receive synchronized parameter values from any of the following items:

- cube view (exploration or visualization)
- websheet
- dimension selector
- synchronization tile

**File**

The **File** control type passes the file name of the uploaded file to the process parameter. This parameter displays only when the **Allow file upload** property is selected. By default, the **Prompt user** is disabled for this parameter.

**Cell**
The **Cell** control type passes a specific cell value based on the selected cube and members that determine the intersection.

For numeric parameters, the following Control types are available:

**Text input**
Presents a box where the user can enter the parameter value. You do not need to specify any **Control detail**.

**List**
Presents a list of values that can be chosen to set the parameter value. You must enter the values that you want to appear in the list in the **Control detail** column, with all values separated by commas.

**Boolean**
Presents binary values that the user can choose. In the **Control detail** column, you can specify the **True label** and **False label**. The default values are `True` and `False`, but you can use `Yes` and `No`, `Yup` and `Nope`, or any labels you want.

**Number range**
Presents a box where the user can enter the parameter value. The value entered is validated against the range that is specified by the **From** and **To** limits in the **Control detail** column. If the value entered is outside the range, the user receives an error message.

**Calendar**
Presents a calendar from which the user can select a date value.

**Cell**
The **Cell** control type passes a specific cell value based on the selected cube and members that determine the intersection.

7. If necessary, enter a **Default** value that is appropriate for the Control type you're using. If you chose to not prompt users for parameter input, the default value is passed directly into the process when it is run.

   Not all Control types require you to specify a default value, some derive the default value as outlined here:

   **Text input**
   Enter the default value that you want to use.

   **List**
   Uses the first value in your comma-separated list as the default value. You can't modify this default value directly.

   **Dimension list**
   After you define the dimension set from which users can pick from in the **Control detail** column, click the **Pick an element** list in the Default column. Pick the member that you want to use as the default value.

   **Boolean**
   After you define a True label and False label, those labels appear in a menu on the Default column. Click the menu and pick the label that you want to use as the default value.

   **Number range**
   Uses the From (lowest) value in number range as the default value. You can't modify this default value directly.

   **Calendar**

   When you specify a calendar Control type, a date picker  is available on the Default column. Click the date picker and pick the date that you want to use as the default value.

8. Click **OK** to save the parameter configuration.

9. While still in edit mode, you can click the button and then click  see how the button behaves and how the process runs.

## Upload files to a database with the action button

Use the Action Button's **Allow file upload** property to upload a file to a Planning Analytics database and run a process to use that file.

**Note:** If you are using TM1 Database 12.5, you can only upload a file to the database if you have the Administrator role in TM1 Database. You cannot upload a file even if you have the Administrator role in Planning Analytics Workspace but not TM1 Database.

This can be helpful if you want to import data from a specific file into a view you are using.

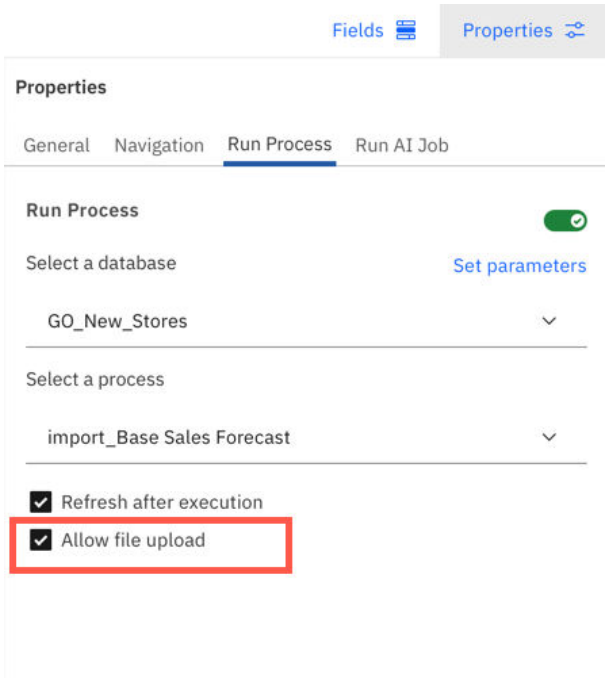
When you configure an action button to run an existing process, selecting **Allow file upload** prompts you to upload a file when you click the action button.

**Note:** The process runs only if the file upload is successful.

To configure an Action Button to upload a file:

1. In **Edit** mode, add an Action Button to your book.

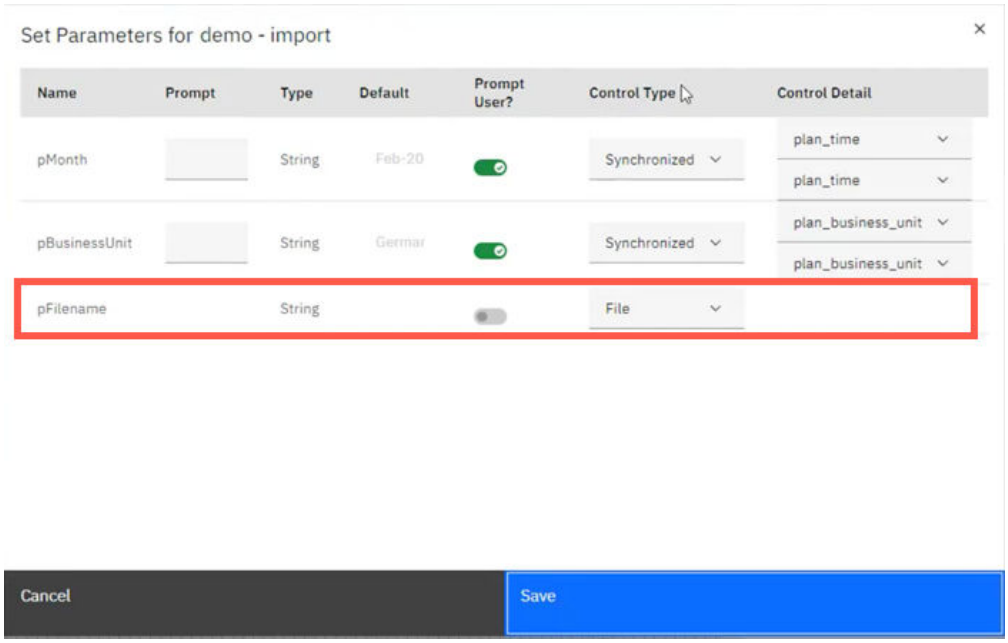
2. Select the Action Button and click **Properties**.
3. On the **Run Process** tab, enable **Run Process**.
4. Select the database on which the process resides.
5. Select the process that you want to run.
6. Select **Allow file upload**.




7. Click **Set parameters** to set the parameter values or configure user prompts that can be answered to set parameter values.

   When you select **Allow file upload**, a **File** control type parameter is added to the parameters. The **File** parameter passes the name of the uploaded file to the process parameter.

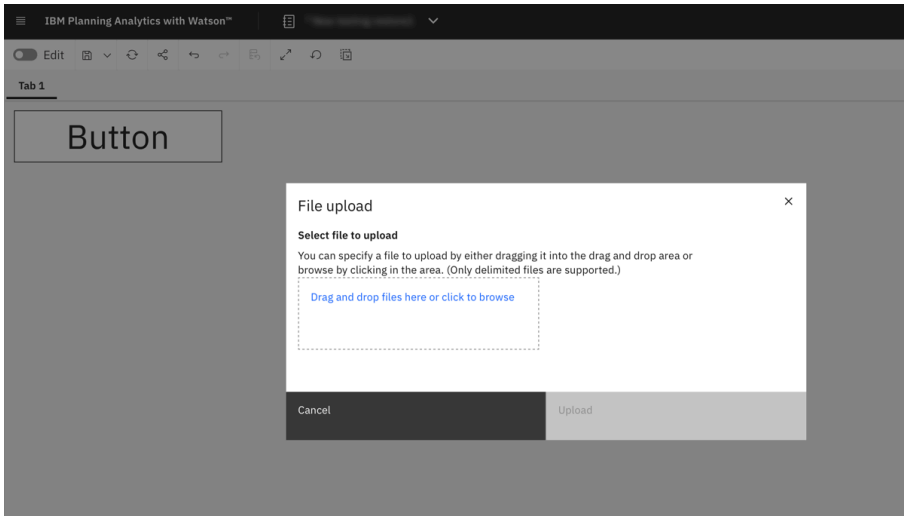
   You can select the **File** control type for the file name parameter in the process. While this is optional, using the **File** parameter is helpful when you want to support any file name as the data source for the process.



**Note:** By default, the **Prompt User** option is disabled for **File** control type.

8. Select **Refresh after execution** if you want to refresh the current book after the process finishes running. The refresh updates any views, web sheets, visualizations, or selectors in the target book.

Now, when you click the Action Button to run the process, a **File upload** prompt displays.




Select a file to upload and click **Upload**. After the file is uploaded successfully, the process starts to run.

# Websheets

You can work with websheets in IBM Planning Analytics Workspace. You can create and work with sandboxes in websheets, and you can combine websheets and views on the same sheet.

A websheet is a Microsoft Excel worksheet (.xlsx file) with TM1 data that you can view in a web browser. Websheets are located in the **Applications** folder in the tree.

Websheets support the accessibility features available in the Firefox browser. However, you should not enable any accessibility features in Chrome when using websheets. Enabling accessibility features in Chrome (chrome://accessibility/) can result in performance degradation. For more details on accessibility features in websheets, see https://www.ibm.com/docs/en/planning-analytics/2.0.0?topic=overview-accessibility-features.

**Note:** When pasting values in a websheet in Planning Analytics Workspace on the Mozilla Firefox browser, a secure connection is required to access the clipboard plug-in.

You can confirm that you have a secure connection if the Planning Analytics Workspace URL is prefixed with the `https` protocol. A secure connection also displays a plain padlock icon on the address bar.

An insecure connection is indicated by the `http` protocol and a padlock icon with a slash through it on the address bar. If you are using a websheet with an insecure connection, you cannot paste values into the websheet.

This issue is unique to pasting in websheets on the Firefox browser and does not occur on other browsers.

## Procedure

1. To add a websheet to a book, you must be in edit mode.
2. Navigate to the **Applications** folder on the server that contains the websheet.

   For example, go to **Planning Sample** > **Applications** > **Planning Sample** > **Bottom Up Input**, and drag **Budget Input** onto the sheet.

3. To link, export (as PDF), or email a websheet, click the websheet then click  **Share**. Complete the details required for your sharing method.
4. To reset data, in the toolbar, click **Reset data**.
5. To rebuild websheets and webbooks, in the toolbar, click **Rebuild websheet** or **Rebuild webbook**.

## Save a websheet to the database

You can save an open websheet to the TM1 database.

### About this task

When you save a websheet to the TM1 database, the websheet is available for quick retrieval from your collection.

A websheet saved in the database is a reference to the original websheet stored in the TM1 database. Any changes made to the original websheet in the TM1 database are propagated automatically to all associated websheets in the database.

### Procedure

1. Click the websheet, then click  to open the toolbar.
2. Click  .

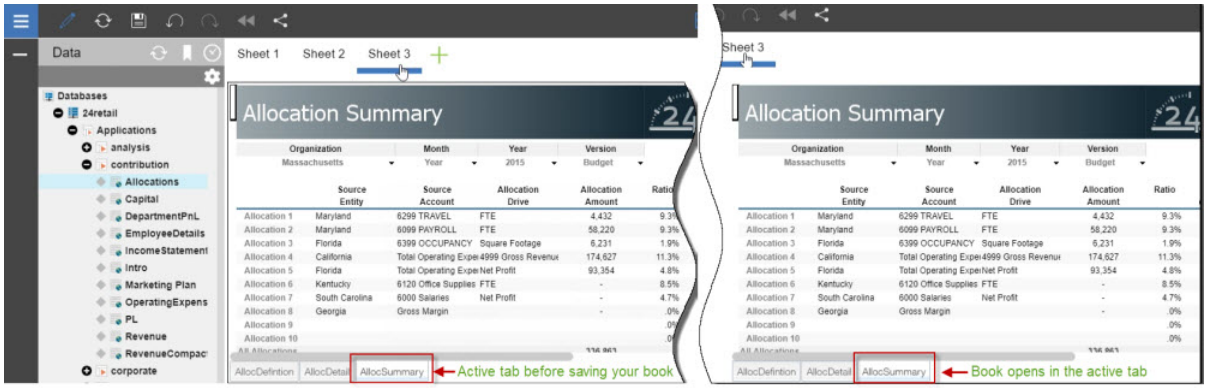
   The first time that you save a websheet, the only option available is **Save As**.
3. Click **Save As**, then provide a name for the websheet, and optionally, tags and a description that are applicable to the websheet.
4. After you initially save a websheet to the content store, you can use the **Save** or **Save as** options.

### Results

The websheet is saved in your personal folder, from which you can open the websheet. You can also click  and then click **Websheets** to access the websheet from your collection. You can't open the websheet directly from the collection, but you can drag the websheet from the collection on to an open Planning Analytics Workspace sheet.

## Open a websheet on the active tab when you save a multi-tab websheet

If you have a websheet with multiple tabs in a book, Planning Analytics Workspace keeps track of the active websheet tab when you save the book. Then, when you open the book in Planning Analytics Workspace later, you are where you left off in your work!




Using Planning Analytics version 2.0.9 and Planning Analytics Workspace version 2.0.46, when you open a book with a websheet in Planning Analytics Workspace, the tab that you saved the websheet with is active.

**Note:** You must be using Planning Analytics Workspace on IBM Planning Analytics version 2.0.9 to take advantage of this feature of websheets in IBM Planning Analytics TM1 Web.

If you don't have Planning Analytics version 2.0.9, the default tab that the websheet was published with (using TM1 Perspectives or Planning Analytics for Microsoft Excel) is active when you open the book in Planning Analytics Workspace.

# Explore scenarios with sandboxes

Sandboxes let you try out different changes to the data before making those changes public to other users and before committing those changes to the base data. Sandboxes are visible only to you.

To use sandboxes, your administrator must set the **UseSandbox** capability to `Grant`. Capabilities can be set in the }Capabilities control cube.

### Procedure

1. To create a sandbox, click the view or websheet to display the toolbar, and then click and click **Create sandbox**.
2. Name the sandbox.
3. Choose whether you want to create a sandbox from the base data or to create a copy of an existing sandbox, and then click **OK**.

   You can now work in the sandbox. You can move between different sandboxes by selecting them from the drop-down list. version 2
4. If you want to discard all data changes you've made in a sandbox, click to reset data values to the sandbox base.
5. When you are satisfied with the data in a sandbox, and you want to commit it to the base, click **Commit data**.
6. To delete a sandbox, follow these steps.

   a) Click the view, and then click .

   b) Select **Delete a sandbox**, select the sandbox that you want to remove, and click **Delete**.

### What to do next

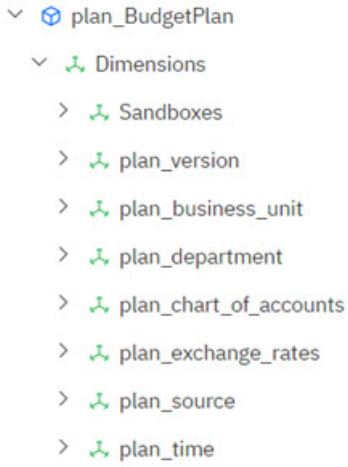
You can set sandboxes to synchronize, so that when a sandbox is set in one view or websheet, other views and websheets update to use the same sandbox. For details, see "Synchronize objects in a book or sheet" on page 593.

## Compare sandboxes

You can create personal scenarios in sandboxes that you can view side by side so that you can compare and calculate the difference between scenarios.

Sandboxes are personal to the creator. If the compare sandbox function is enabled, and the Use Sandbox capability is granted by your administrator, you can use sandboxes as a virtual dimension in a view. An individual sandbox is treated as member in the virtual dimension.

You can see the **Sandboxes** dimension in the **Dimensions** branch in the tree, and as a tile that you can drag from the bench  in a view. You can drag the **Sandboxes** dimension onto a row, column, or onto the context area.




### Example

For example, suppose you have two sandboxes, Best case, and Worst case. You can display these two sandboxes next to each other in nested columns, and then calculate the variance, as shown in the following example.

| | Jan | | |
|---|---|---|---|
| | Best case | Worst case | Variance |
| Channel Total | 221.00 | 200.58 | 20.42 |
| Retail | 200.00 | 184.02 | 15.98 |
| Internet | 15.00 | 11.04 | 3.96 |
| Distribution | 6.00 | 5.52 | .48 |

You can compare sandboxes by doing the following steps:

1. Drag the **Sandboxes** tile from  , and nest it with the Month tile on columns.

**Tip:** You can also drag **Sandboxes** from the **Dimensions** branch in the tree.

2. Right-click, or tap and hold on the Base member column heading and select **Hide**.
3. Create a variance calculation by selecting the Best case and Worst case column headers, right-clicking, and select **Create calculation**.
4. Name the calculation, choose **Best case - Worst case**, and click **OK**.

### Limitations

You cannot spread data across multiple sandboxes.

You also cannot add members to the **Sandboxes** dimension. Sandboxes are added to the dimension when you create a new sandbox.

You can create unnamed sets for the sandbox dimension, and apply them to the current view, but you can't name and save sets.

### Enabling the sandbox compare function

To enable the sandbox compare function, ask your TM1 administrator to add `EnableSandboxDimension=T` to the `tm1s.cfg` file of the TM1 database that you are using. For more information, see EnableSandboxDimension on IBM Knowledge Center.

## Use the Planning Analytics AI assistant to interact with books using natural language

The Planning Analytics AI assistant lets you perform a wide variety of actions on books using simple natural language commands.

Planning Analytics AI assistant supports English, German, French, and Spanish. If you're using Planning Analytics Workspace in any other language the Planning Analytics AI assistant defaults to English.

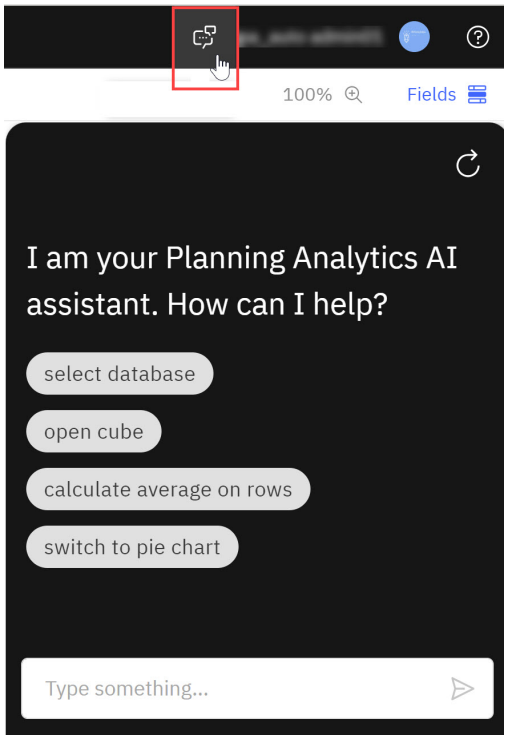
The Planning Analytics AI assistant is available only when working directly in a book. The assistant is not available when you open a book from a plan or application, nor is it available from a modeling workbench.

**Important:**

Planning Analytics AI assistant requires a separate license purchase, it is not included in a standard Planning Analytics Workspace license. To purchase a license, contact your IBM sales representative or go directly to your IBM account.

To enable Planning Analytics AI assistant on Planning Analytics Workspace Local, you must set the **`ENABLE_WATSONX_ASSISTANT`** and **`WATSONX_ASSISTANT_CONFIG`** configuration parameters in the `paw.ps1` (Windows) or `paw.env` (Linux) configuration file.

To use the assistant, open a book and click the **Planning Analytics AI assistant** icon on the toolbar, right next to your user name.

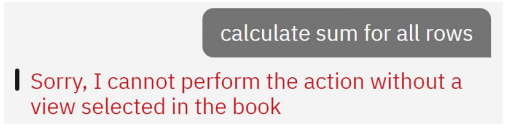



Type a natural language command or instruction in the Planning Analytics AI assistant. There are no hard and fast rules about how you issue a command, as long as the command includes a single action that you want to perform and (in most cases) the subject of the action. For example, if you want to swap the rows and columns in an exploration, all of these commands will work:

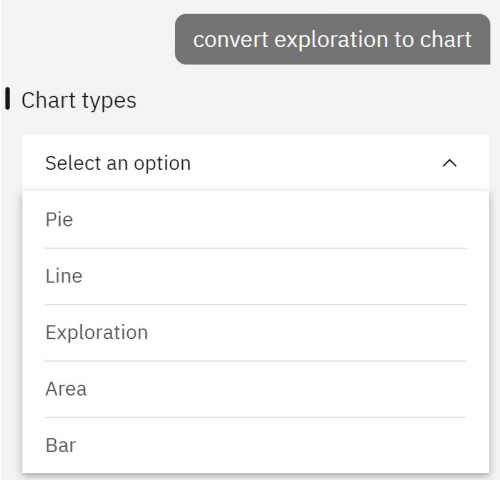
- `swap rows and columns`
- `switch rows`
- `exchange rows and columns`
- swap (notice there is no subject specified here)

To reuse a previous command from the current session, place the cursor in the Planning Analytics AI assistant text entry box, then use the Up and Down keyboard keys to cycle through recent commands. When the desired command is in the text box, press Enter to use the command.

Some commands require that an object or widget on a book be selected before typing the command, others require that the book be in Edit mode before the command can be executed. Some commands are applicable only to explorations, while others are applicable only to visualizations. In these circumstances, Planning Analytics AI assistant alerts you to the requirement.




When your command is ambiguous or cannot be completed in the current context, Planning Analytics AI assistant prompts you for clarification. For example, the command "`convert exploration to chart`" is somewhat ambiguous, as it does not specify a chart type. The assistant prompts you to specify which type of chart you want and then completes your request.




**Tip:** If your command includes a reference to a specific object or artifact, it's helpful to enclose the item name in double quotes. This helps Planning Analytics AI assistant determine your intent and identify the item you want to work with.

You can use Planning Analytics AI assistant to change the configuration of a view, open a view on a different database, open a dimension or set, perform member calculations, suppress zeroes, and more.

Remember, a command specifies a single action; you can't issue a compound command to perform multiple actions. For example, the command "`open the Revenue cube and suppress zeroes`" won't work.

Here's a "cheat sheet" to help you start using Planning Analytics AI assistant. It shows all the actions that you can perform with the assistant.

Notice that there are multiple ways to express most commands. The assistant recognizes synonyms for most actions, so you're not limited to one specific command construct. Experiment with commands, knowing that the assistant will prompt for clarification if a command is ambiguous.

| **Action** | **Definition of action** | **Command examples** | **Needs object/ widget selected?** |
|---|---|---|---|
| **Database interaction** | | | |
| List | Show all the specified objects on a database. You can list cubes, dimensions, sets, and views. | `List cubes`<br>`List dimensions` | No |
| Open | Open one of these Planning Analytics objects:<br>• Cube (default view)<br>• Exploration (view)<br>• Dimension<br>• Hierarchy<br>• Set | `Open dim "Org"`<br>`Open "Asset Life" view`<br>`Show the "Marketing" set`<br>`Edit the "Employee" hierarchy` | No |
| Search | Search for TM1 objects such as cubes and dimensions within your servers | `Search for 'Budget_Allocation'` | No |

| Action | Definition of action | Command examples | Needs object/ widget selected? |
|---|---|---|---|
| | | `Search for 'Budget_Plan' in 'Planning Server'`<br>`Search for dimension 'Budget' in 'Planning'` | |
| Select database | Set the search context for the Open actions in the book. | `Select database 24Retail and Planning Sample`<br>`Work with "Planning Sample"`<br>`Set "24Retail" as database` | No |
| Show databases | Display the databases currently being searched. | `What is my search context?`<br>`What is the current db?` | No |
| **Generative AI** | | | |
| Explore data | Use the Data explorer. | `Explore my data on 'my_server' with this generic query...`<br>`Explore my data on 'this_cube' on 'my_server' with this specific query...`<br>If you do not specify a server, the assistant uses the current search context.<br>If only one database is available, the assistant proceeds automatically, but if multiple databases are found, you are prompted to choose one. | No |
| Impact analysis | Display an AI-generated impact analysis for an exploration, which helps you to determine the impact of data fields on each other as well as identify and quantify relationships across the data. | `Impact analysis`<br>`Analyze view`<br>`Analyze` | Yes |
| Outlier analysis | Display AI-generated outlier analysis for an exploration, which identifies and explains outliers found within the exploration. | `Run outlier analysis`<br>`Detect outliers` | Yes |

| Action | Definition of action | Command examples | Needs object/ widget selected? |
|---|---|---|---|
| | | `Outlier analysis` | |
| Show chart insights | Display AI-generated insights that help you interpret data in a visualization. | `Show insights`<br>`Explain visualization` | Yes |
| **Book interaction** | | | |
| Cancel | Cancel the current operation. | `Cancel`<br>`Quit` | No |
| Copy | Copy the selected widget. | `Copy`<br>`Duplicate` | Yes |
| Refresh | Refresh a view. | `Refresh`<br>`Reload view` | Yes |
| Redo | Redo a book action. | `Redo` | No |
| Switch visualization | Switch the visualization to a different type. | `Switch to pie chart for "sales"`<br>`Convert to chart`<br>`Switch to exploration` | Yes |
| Undo | Undo a book action. | `Undo` | No |
| **View manipulation** | | | |
| Collapse | Collapse a consolidation. | `Collapse "Total year"` | Yes |
| Create sandbox | Create a sandbox. | `Create sandbox "prev_2024"`<br>`Sandbox` | Yes |
| Display zeroes | Display zeroes previously suppressed. | `Display zeroes on rows`<br>`Show 0 rows`<br>`Display zeroes on rows` | Yes |
| Expand | Expand a consolidation. | `Expand "Total year"`<br>`Drill "Canada"` | Yes |
| Hide | Hide a member(s) from rows or columns. | `Hide "Jan", "Feb"` | Yes |

| Action | Definition of action | Command examples | Needs object/ widget selected? |
|---|---|---|---|
| | | `Don't show the "Overhead" member` | |
| Keep | Keep only the specified member(s) on the view. | `Keep "Jan", "Feb"`<br>`Members to keep: "Washington", "California"` | Yes |
| Move | Move a dimension/hierarchy to a slot in a view. | `Move "Version" to rows`<br>`Transfer "Channel" to bench`<br>`Relocate "Model" to columns` | Yes |
| Perform member calculation | Perform calculations on specified members.<br>The supported calculations are:<br>• Absolute value<br>• Aggregate<br>• Average<br>• Division<br>• Maximum<br>• Median<br>• Minimum<br>• Multiplication<br>• Percent change<br>• Percent of<br>• Percent parent<br>• Percent total<br>• Rank<br>• Subtraction<br>• Summation | `Calculate average for members "Jan", "Feb"`<br>`Create maximum for members "Q1", "Q2"`<br>`Calculate minimum for members "Q1", "Q2"` | Yes |
| Slice | Switch the context of a view to a specific member. | `Slice on "Canada"`<br>`Change context to "Budget"` | Yes |
| Summarize all | Create a calculation of all rows or columns. | `Calculate average on rows`<br>`Show sum on columns`<br>`Aggregate rows as "MyTotal"` | Yes |

| Action | Definition of action | Command examples | Needs object/ widget selected? |
|---|---|---|---|
| Suppress zeroes | Suppress zeroes on the view. | `Suppress zeroes on rows`<br>`Hide 0 rows`<br>`Remove cols with value 0` | Yes |
| Swap | Swap rows and columns in a view. | `Swap`<br>`Switch`<br>`Flip rows and columns` | Yes |
| Switch set | Switch the set of dimensions in the rows or columns of an exploration. | `Switch to set 'Actual'`<br>`Switch set 'Budget'` | Yes |
| Switch to sandbox | Switch to an existing sandbox. | `Switch to "Base" sandbox`<br>`Change sandbox to "MyProjections"` | Yes |
| Unhide all | Display all hidden members. | `Unhide all`<br>`Show all rows`<br>`Uncover columns` | Yes |

## Snap commands

Snap commands are simple commands that you can use to perform tasks quickly.

Click a view, then click  and type a snap command.

You can type snap commands in full or use abbreviations. For example. **swap** and **sw** are equivalent. Snap commands aren't case-sensitive and you don't need to use exact spelling - `beljium` selects `Belgium`.

You can use these snap commands:

**Select**
Select members, sets, and levels in dimensions.

Type the names that you want to select in the snap command field that is separated by a `,` (comma), or `and`. Select is the default snap command, so you do not need to type `select`.

For example, type `germ,mark` to select the Germany and Marketing members.

Select doesn't apply to members on the bench.

**Swap**
Swap rows, columns, dimensions in the context area. For example, to swap the versions and time dimensions, type `sw versions,time`. Typing `swap` on its own swaps the rows and columns.

**Find**
You can find members on rows, columns, or both.

`Find europe` finds the first result. To move through the results, type `find`. To go back, type, `find previous`.

Find is a powerful snap command, for example, typing `find r (color red or color blue) and size large` finds members on the rows, with an attribute value for color that is red or blue, and an attribute value for size that is large.

**Hide**

Hide named members in dimensions on the rows or columns.

To hide members that are named US and Feb-2004, type `hide US, Feb-2004`.

**Unhide**

Show or unhide members.

Type `unhide r` to unhide rows. To unhide all, type `unhide`, or `unh`.

To unhide everything on the time hierarchy, type `unh time`.

**Keep**

Keep members in dimensions on the rows or columns, while you hide the other members.

For example, type `keep 2014, 2015`.

**Zero**

Enable zero suppression.

To enable zero suppression on just rows type `zero rows`, and for columns, type `zero columns`.

To clear all zero suppression, type `zero off`.

**Totals**

Show leading or trailing totals, or hide totals.

For example, in a time dimension, you could have Q1, Jan, Feb, Mar, which has a leading total. Typing `Totals trailing columns` would change the time dimension to: Jan, Feb, Mar, Q1.

Type `t -h` to hide all totals.

**Level**

Expand levels in a hierarchy to a specific depth. If you don't specify a level, the leaf level is expanded.

You can also expand numbered levels, and levels on rows, columns, or both. For example, type `level rows 2` or `level rows`.

**Drill**

Drill down on members.

For example, to drill down on a member named Georgia, type `Drill Georgia`.

Resetting the drill state clears drills. To reset the drill state for an Organization hierarchy, type `Drill reset Org`, or `d r org`.

To remove all of the drills, type `drill reset`.

**Sort**

Sort labels or values in ascending or descending order. You can choose to sort by hierarchy, or to break the hierarchy.

You can sort a named hierarchy, and for labels, you can sort by rows and columns. You don't need to specify label or value, unless there is an ambiguity.

**Sort lab asc rows**

Sorts labels in ascending order for rows, without breaking the hierarchy. You can also type this in natural language: `sort label ascending rows`.

**Sort lab des br both**

Sorts labels in descending order for both rows and columns, and breaks the hierarchy.

**Sort val des br Sales**

Sorts values in descending order, breaking the hierarchy, under the Sales member. You must specify a member to sort for values. In natural language, this is: `sort value descending breaking Sales`.

**sort -x**

Clears all sorting. To clear sorting on the months hierarchy, type `sort clear months`.

### How the snap command searches

IBM Planning Analytics Workspace searches for member names, captions, and aliases, named sets, and named levels (levels with default names are ignored). If IBM Planning Analytics Workspace finds more than one member, they are listed in order of relative strength of the matches. More weight is given to precise matches than tentative matches.

# Access your data and other objects quickly

You can quickly access your data and other objects in IBM Planning Analytics Workspace.

In Planning Analytics Workspace Classic, you can add a view, a visualization, or objects to a collection to reuse them in a book. You can access recently used objects in the navigation tree. You can bookmark items in the navigation tree for later retrieval.

In Planning Analytics Workspace new experience, you can pin items.

## Pin items

When you create a useful view, visualization, selector, image, video, or other item, you can pin the item for use in other books or websheets. The pinned items are added to the **Pins** tab.

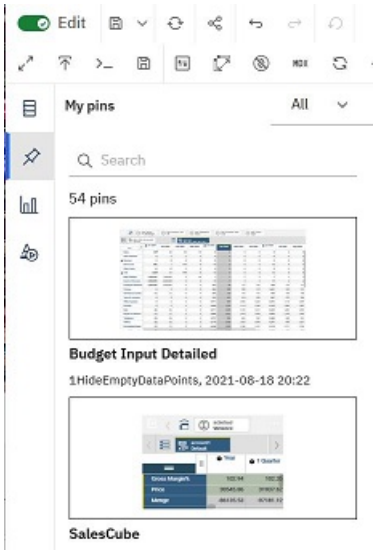
You can pin items in the edit mode or the consumption mode if pinning is enabled for this mode. To enable pinning for the consumption mode, use the book property **Allow pinning in consumption**. For more information, see "Enable pinning in consumption mode" on page 442.

To retrieve an item from **Pins**, you must be in the **Edit** mode.

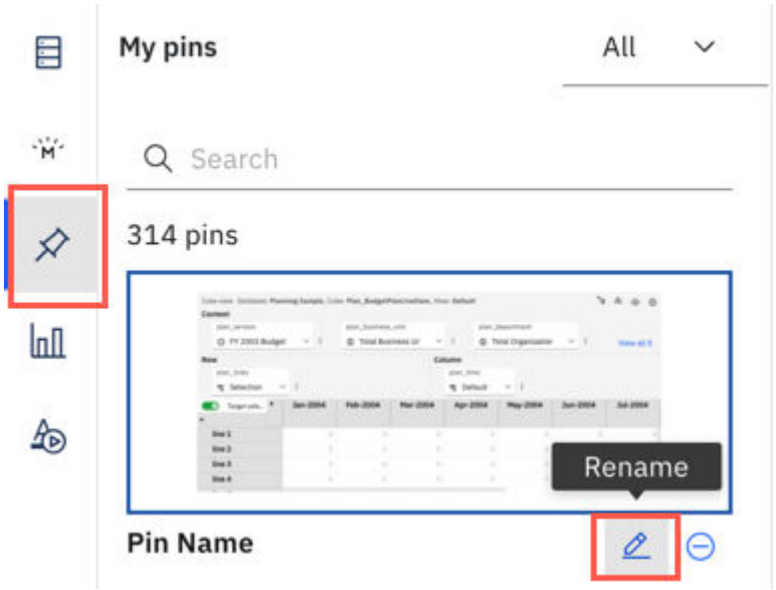
### Procedure

1. In a book or view, click anywhere on the item that you want to pin, and click the pin button in the toolbar.

   The item is added to the **Pins** tab.




2. To retrieve an item from the pins collection, ensure that you're in the **Edit** mode. Click the **Pins** tab, and search for the pinned item. Click or drag the item to add it to the canvas.
3. Once the item is pinned, you can rename it. Click the **Rename** icon next to the name of the pinned item. Enter the new pin name and click **Done** or press Enter.




## Access recently used items

IBM Planning Analytics Workspace saves a list of recently used items so that you can quickly locate those items in the navigation tree.

### Procedure

1. Click  on the navigation tree.

   **Tip:** If the tree is not visible, click  .
2. Click the item that you want to locate in the **Recently visited** list.
   The item is selected in the navigation tree.

## Bookmark items

You can bookmark items in the data tree for easy retrieval.

### About this task

You can bookmark individual items, but not categories of items. For example, you can bookmark a specific cube, but not the entire Cubes category.

### Procedure

1. To bookmark an item, right-click the item in the data tree, then tap **Bookmark this**.

   **Tip:** If the data tree is not visible, click  .
2. Click  .
3. Click the bookmarked item that you want to locate in the data tree.
   The item is selected in the data tree.

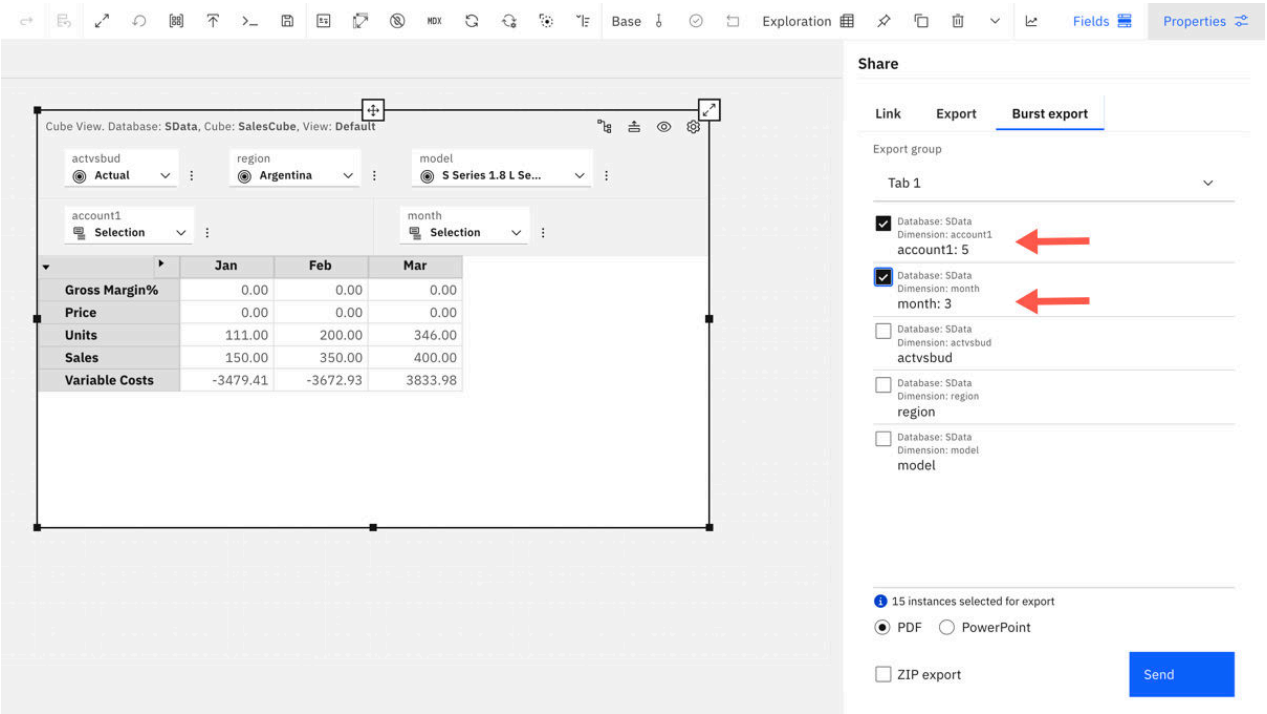
# Burst export to PDF or PowerPoint

You can slice a book in Planning Analytics Workspace and burst export it to PDF or PowerPoint by using hierarchy synchronization. With this **Burst export** feature, you can automatically print each member of the set for a selected hierarchy to a PDF or PowerPoint file.

To use the **Burst export** feature, follow these steps:

1. In **Edit** mode, select the view and enable hierarchy synchronization (**Properties > Synchronization > Synchronize hierarchies**).
2. Click **Share** in the toolbar and go to the **Burst export** tab.
3. Select the group that you want to export under **Export group**.
4. Select the hierarchies that you want to export.

   **Note:** When you select a hierarchy, the number of members in that hierarchy displays under the hierarchy name. If you select more than one hierarchy, the total number of instances that export are calculated based on all the data combinations for the selected hierarchies. You cannot export more than 100 instances.
5. Select the file type that you want to export to. If you want to export individual files for each instance, select **ZIP export**.




# Export to Excel

You can export a cube view as a Microsoft Excel spreadsheet.

## About this task

There are some size limitations that you need to be aware of. If you reach any of these limitations, you receive an error message.

**Maximum number of rows**

1048576

**Maximum number of columns**

16384

**Maximum number of cells**
8 million

**Maximum number of view instances**
100

The total number of instances is determined by multiplying the number next to each option. If the number of instances is greater than 100, try changing your selection.

If the view includes attribute values on either the rows or columns, the attributes and the relevant headers are exported. The header cell contains the attribute name. Note that since attributes can exist on both rows and columns, there may be a conflict as to which attribute name should show up in the header cell. In this case, attributes on the rows take priority.

Where there is a parent-child relationship between the members, the rows are indented even if the rows aren't indented in Planning Analytics Workspace.

Export to Excel is not available from an iPAD.

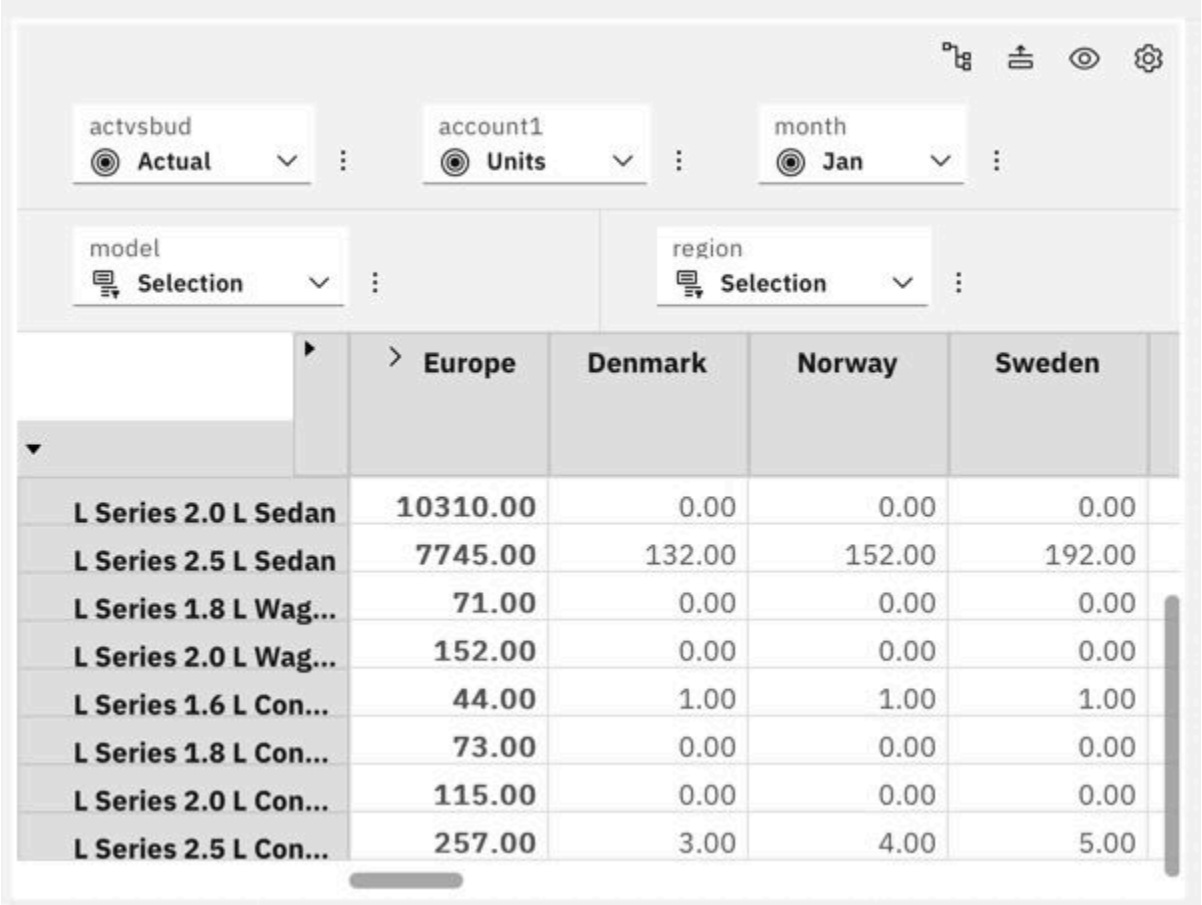
## Procedure

1. Click the cube view that contains the data that you want to export, then click on the toolbar.
2. Optional: Select which parts of the view that you want to export, based on hierarchy. Click on the **Burst based on the following hierarchies** and select the hierarchies you want to export.

   An option is shown for each dimension hierarchy that appears in the Context bar. If you select nothing, then the selection of data that is shown in the view is exported.
3. Choose whether you want to export all sheets to a single file, or to multiple files. If you didn't choose to burst based on a hierarchy, only one sheet can be exported.
4. Click **Export** to begin the export process.

### Example

Take as an example a view that shows models on the rows, and regions on the columns. On the context bar, **`Variance`** is selected in the **`actvsbud`** tile and **`Units`** is selected in the **`account1`** tile. If you don't select a hierarchy to burst to, the view is exported as selected, with no additional sheets. But if you select a hierarchy, all possible combinations of views are exported.




For example, the **actvsbud** hierarchy has the following three items:

- Variance
- Actual
- Budget

If you select **actvsbud** hierarchy, data is exported to three sheets: Variance, Actual, and Budget.

The **account1** hierarchy has three items at the same level:

- Units
- Sales
- Variable costs

If you select **account1** hierarchy, data is exported to nine sheets:

- Variance_Units
- Variance_Sales
- Variance_Variable costs
- Actual_Units
- Actual_Sales
- Actual_Variable costs
- Budget_Units
- Budget_Sales
- Budget_Variable costs

# Export data from a cube or view

You can export data from a cube or view to a `.csv` file. The exported file is saved to your local `Downloads` directory.

## About this task

You can initiate a data export from several places in the book **Databases** tree:

**Database**
Right-click a database, then click **Import/Export** > **Export data**.

**Cubes node for a database**
Right-click the **Cubes** node for a database, then click **Export data**.

**Cube**
Right-click a specific cube, then click **Import/Export** > **Export data**.

**View**
Right-click a specific view, then click **Export data**.

## Procedure

1. Start the data export from one of the available locations in the book **Databases** tree.
2. If necessary, select the **Cube** from which you want to export data. Selecting the cube is required only if you initiate an export from a database or from the **Cubes** node on a database.
3. By default, the name for the exported `.csv` file is the same as the source cube. If you want to use a different name, enter it in the **File name** box. You do not need to include a file extension.
4. Choose whether you want to export all cube/view data or want to refine the scope of the data that is exported.
   - Click **Export** to export all cube or view data.

     By default, an export does not include consolidated values, rule-derived values, or zero/blank values.

     If you initiated data export from a database, **Cubes** node, or individual cube, all data from the source cube is exported.

     If you initiated data export from an individual view, only the data included in the source view is exported.
   - Click **Next** to set options that define the data that is exported.
5. Optionally, select a **View** as the basis for the export.

   When you select a view, the **Dimension elements** grid updates to reflect the view configuration.
6. Use any combination of the **Hierarchy**, **Set**, and **Member** menus to further define the scope of data to export for each dimension.

   For example, you can:

   - specify only a hierarchy to include all members of the hierarchy in the data export.
   - specify a hierarchy and set to include all members of the set in the data export.
   - specify a hierarchy, set, and single member to include only the selected member in the data export.
7. If you want to filter the numeric values that are included in the export, click the **Numeric values operator** menu and select an expression. Enter the values that are required to evaluate the expression.
8. If you want to filter the string values that are included in the export, click the **String values operator** menu and select an expression. Enter the values that are required to evaluate the expression.
9. By default, an export does not include consolidated values, rule-derived values, or zero/blank values. If you want to include any of these types of values in an export, clear the checkbox next to the appropriate **Skip** option.

10. Click **Export**.

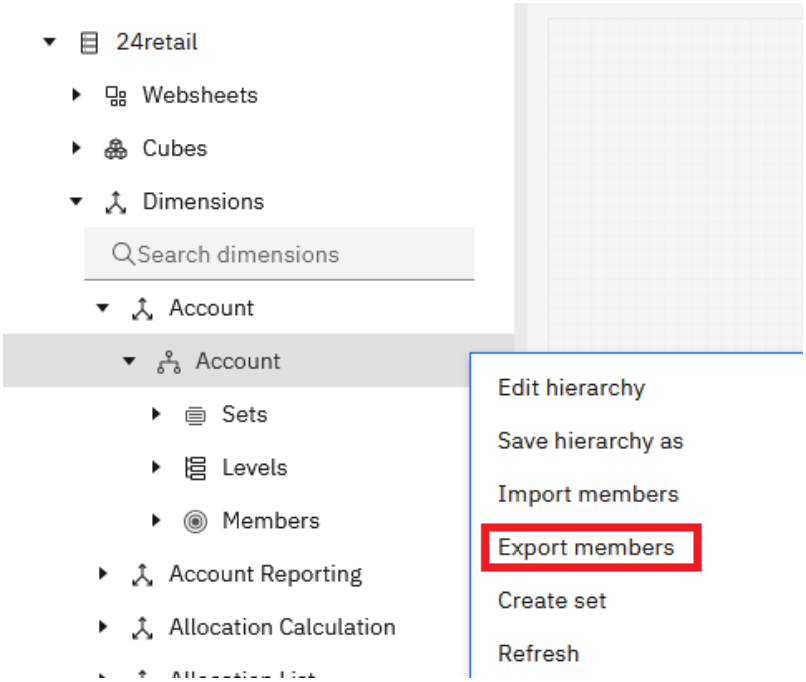
    The exported data is saved in a `.csv` file in your browser **Downloads** directory, using the name specified in step 3.

    You can save your export settings to the view, so when you export data from the same view at a later date, your export settings are automatically selected. If you make additional changes to your export settings, you are prompted to save your changes. All users with read access to a cube can save export settings for their own private views, while users with the **Administrator** role can save export settings for public views.

## Export a dimension or hierarchy

You can export a dimension or hierarchy to a `.csv` file. The exported file is saved to your local `Downloads` directory.

To export, right-click the dimension or hierarchy in a book **Databases** tree, then click **Export members**.




You can also click the **Export members** icon in the dimension editor to export a dimension.

When you export a dimension or hierarchy, the resulting `.csv` file includes the following information about each member:

- Name
- Parent-child relationship
- Type
- Weight
- Defined attribute names and values

  Attribute names are prefixed with either ATTR_N or ATTR_S when they are exported, depending on the attribute type (numeric or string).

# How to parameterize an MDX query in IBM Planning Analytics Workspace

You can parameterize MDX queries in IBM Planning Analytics Workspace.

The default MDX generation uses a query state, which consists of the sets for each hierarchy, member selections for hierarchies on titles, and any applied operations (for example, expand, collapse, sort, etc). The query state generates an MDX query to retrieve the corresponding axis and cell data from the TM1 database.

A parameterized MDX involves the use of an MDX template. MDX templates allow you to override how the query state is used to generate the MDX. An MDX template contains the following:

- Variables that can be substituted during MDX generation.
- Variable declarations which define the members or sets in the query state which are substituted for the variables in the MDX template.

Supporting parameters allows for more control over the MDX, while allowing the view to respond to the specified member and set synchronization events based on how the template is defined.

The following variables can be defined in an MDX template:

**Set variables**
Set variables define a reference to a set for a hierarchy defined in the query state.

For example:

```
{
      "Name": "<0>",
      "HierarchyID": "[plan_time].[plan_time]",
      "Type": "SetVariable"
    }
```

**Slicer member variables**
Slicer member variables define a reference to the selected member for the query state.

For example:

```
{
      "Name": "<2>",
      "HierarchyID": "[plan_version].[plan_version]",
      "Type": "SlicerVariable"
    }
```

**Set member variables**
Set member variables define references to members of a set at a specified index.

For example:

```
{
      "Name": "<1>",
      "HierarchyID": "[plan_chart_of_accounts].[plan_chart_of_accounts]",
      "Type": "SetMemberVariable",
      "Index": 0
    }
```

The following is an example of a complete MDX template:

```
{
  "MdxTemplate": "SELECT {<0>} ON 0, {<1>} ON 1 FROM [plan_BudgetPlan] WHERE (<2>,
[plan_business_unit].[plan_business_unit].[10110])",
  "Variables": [
    {
      "Name": "<0>",
      "HierarchyID": "[plan_time].[plan_time]",
      "Type": "SetVariable"
    },
    {
      "Name": "<1>",
      "HierarchyID": "[plan_chart_of_accounts].[plan_chart_of_accounts]",
      "Type": "SetMemberVariable",
```

```
      "Index": 0
    },
    {
      "Name": "<2>",
      "HierarchyID": "[plan_version].[plan_version]",
      "Type": "SlicerVariable"
    }
  ]
}
```

## How to parametrize an MDX query

To parametrize an MDX query, complete the following steps:

1. In the cube viewer, click the edit MDX icon MDX to access the MDX.
2. Click the MDX template variables icon </>. The **MDX parameters** dialog box opens.
3. To add a parameter, click the **Add parameter** button and set the following options:
   a. **Name**: The parameter id. The name can only consist of numeric values.
   b. **Dimension**: The dimension to define in the parameter.
   c. **Hierarchy**: The hierarchy to define in the parameter.
   d. **Type**: The type of variable to define in the parameter. You can choose a **Set**, **Set member**, or **Slicer** variable.
   e. **Index**
4. To add the current parameter to the list and create another parameter, click **Add**. To add the current parameter to the MDX and close the **MDX parameters** dialog box, click **Add and close**.
5. Click **Done**.

## Known limitations

There are a few current known limitations with MDX templates:

- Apart from the defined variables, MDX template views are read-only. They do not respond to events that change the structure of the query (for example, pivoting, expand, collapse, sort, etc).
- MDX template views only respond to member and set synchronization events they are parameterized to respond to.
- Metadata change recovery is only applied to sets and members referenced by variables.

# Chapter 4. Use applications and plans to organize work

Applications and plans let you organize logically related Planning Analytics Workspace assets such as books, views, and websheets in containers.

**Note:** Applications and plans are not available in Planning Analytics Trial.

An application contains related assets that are grouped in sections. These sections might reflect the structure of your organization, planning and budgeting requirements, or any other relevant grouping of assets. While an application contains logically related assets, there are no implied or required actions associated with the assets or sections in an application.

A plan contains assets that are grouped in tasks. These tasks can represent discrete tasks or contributions that must be completed in a planning or budgeting process. While tasks can be ordered in a plan, there is no requirement for contributors to complete the tasks sequentially; they can be completed in any order. However, a plan administrator can define task dependencies, so that a task becomes available to users only upon the completion of a different task. Tasks can also be assigned a due date for contributions.

Access to sections in an application or tasks in a plan are controlled by Planning Analytics Workspace user group assignments. When a user group is assigned to a section or task, the security for the contained assets is updated to include the user group.

**Important:** Application and plan security is applied only to Planning Analytics Workspace user groups and roles. There are no links or dependencies to TM1 object security. It is the responsibility of the Planning Analytics Workspace administrator to ensure that the Planning Analytics Workspace user groups have the requisite TM1 security permissions to view and edit assets.

When you log in to Planning Analytics Workspace, your applications and plans are visible on the **My Applications** tab at the bottom of the home page. Applications are identified by the icon, while plans show the icon.

Administrators can see all applications and plans on the home page, regardless of state or group assignments. Modelers, analysts and consumers can see open applications or plans in which they are a member of at least one of the user groups assigned to the included assets.

## Applications

An application contains related assets, such as books, views, and websheets, that are grouped in sections. These sections might reflect the structure of your organization, planning and budgeting requirements, or any other relevant grouping of assets. While an application contains logically related assets, there are no implied or required actions associated with the assets or sections in an application.

Applications are created by Planning Analytics Workspace administrators, analysts, and modelers. The application creator can give other users owner permissions for the application. When you log in to Planning Analytics Workspace, the applications that you can use appear at the bottom of the home page on the **Your Applications** tab. Applications are identified by the icon.

You can view the details of an application from either the home page or the **Applications and Plans** page. The details page shows the status, announcements, sections, and assets for an application.

While the details page is largely a 'read-only' snapshot of an application, you can click any asset to open the full contribution page for the application.

To view the details of an application, use one of the following methods:

© Copyright IBM Corp. 2009, 2025

- If you are a user with the **Full control** permission, click the **Options** icon ⋮ on the application tile, then click **Details**. You can also click the **Options** icon ⋮ for the application on the **Applications and Plans** page, then click **Details**.
- If you are a consumer user, click the application tile on the Home page to open the plan details. You can also click the application name on the **Applications and Plans** page to view application details.

# Create an application

Create an application to organize your books, views, and websheets

You must be a Planning Analytics Workspace administrator, analyst, or modeler to create an application. However, when you create an application you can assign one or more co-owners to the application. A co-owner has full control over the application.

Any asset you want to include in an application must reside in the Planning Analytics Workspace Shared folder.

There are three parts to application creation:

1. Define an application and assign co-owners
2. Add sections and contributors
3. Optional: Add announcements

## Define an application and assign co-owners

Create an application to organize assets used in your processes to provide guidance to contributors.

### Procedure

1. On the Planning Analytics Workspace **Home** page, click **Applications and Plans**.
2. On the **Applications and Plans** page, click **Create**, and then click **Application**.

   You can also create an application directly from the **Home** menu ☰. Click the Home menu, then click **New** and select **Application**.
3. On the **Create an application** page, under **Application details**, enter a name and description for your application. The description you provide is visible on the application tile on the Planning Analytics Workspace **Home** page.

   Optional: Use the **Upload files** option to add one or more files to the application overview. Any attachment you add here is available to users as a link immediately following the application name and description.
4. Under **Image**, you can add an image to your application. You can upload an image by clicking in the image box or dragging an image onto the image box.
5. Under **Co-owners**, you can select users that will be co-owners on this application. Co-owners have full control over the application and can modify all aspects of the application.
6. Click **Create**.

   If you have more than one application to create, you can duplicate the application that you just created. On the application page, click **Application actions** and select **Save as**. Rename the duplicate and click **OK**. All details such as the co-owners, images, and attachments are copied over to the duplicated item.

### What to do next

You can now add sections and contributors as well as announcements to the newly-created application.

You can also make the application available to other users by switching the **Application status** toggle to **Open**.

## Add sections and contributors to an application

You can group related assets such as books, views, and websheets in sections of an application. The sections in your application might reflect the structure of your organization, planning and budgeting requirements, or any other relevant grouping of assets.

### Before you begin

You can add sections to an application after you have defined an application and assigned co-owners.

### Procedure

1. On the application page, click **Add a section** on the **Application sections** tab.
2. Enter a name and description for the section.
3. Optional: Use the **Upload files** option to add one or more files to the section. Any attachment you add here is available to users as a link in the section.
4. Under **Tags**, follow these steps to add one or more color-coded tags for the section:
   a) Click into the **Tags** field, and click **Create new tag**
   b) In the **Name** field, enter a label for your tag.
   c) Choose a color for your tag.

      Select from the colors that are available in the dialog box, or click **Custom color picker** and select a custom color. The color picker is different in different browsers.
   d) Click **Create**.
   e) Expand the **Tags** field, select the tag that you just created, and click **Save** to add it to the section.

      Any tags that you create are saved to the application. This means that all sections in an application can share the available tags. Sections can't access tags from other applications.
   f) Click the **Edit a tag** icon next to a tag to edit or delete the tag.

      **Important:** When you delete a tag, the tag is removed from all sections in the application that use the tag.
5. Under **Assets**, select the assets from the **Available** section that you want to include in the current application section.

   The **Assets** tab shows the books, views, and websheets that are available to add to your application section.

   -  - Books
   -  - Views
   -  - Websheets

   You can search asset names or filter by asset type to narrow the list of assets.

   The assets are added to the **Selected** list in the order you select them. You can click and drag assets to different positions to change the ordering.

   **Tip:** In Planning Analytics Workspace 2.0.94 and later versions, you can select a tab to open by default in book assets.
6. Under **Contributors**, select the user groups that need access to the section.

   The **Add groups** page shows the existing user groups that you can assign to the section assets. You can select any combination of existing user groups or the Everyone virtual user group. The Everyone group assigns all users in your environment to a section.

   When a user group is assigned to a section, the security for the section assets is updated to provide access for the user group.

   **Important:** Application security is applied only to Planning Analytics Workspace user groups. There are no links or dependencies to TM1 object security. It is the responsibility of the Planning Analytics

Workspace administrator to ensure that the Planning Analytics Workspace user groups have the requisite TM1 security permissions to view and edit assets.

7. Click **Add section** to finish adding the section to your application.
8. Optional: Repeat steps 1 through 5 for each additional section you want to add to the application.

   You can also click **Duplicate section** to create a similar section.

   When you duplicate a section, details such as the **Assets**, **Contributor** groups, and attachments are copied over to the duplicated item. The section name in the duplicate section is different from the original. The duplicated section name includes the original name and appends an underscore and number at the end of the section name.

### What to do next

You can add an announcement to your application to provide general instructions and other details for your contributors.

To view the section summary page, on the **Application sections** tab, expand the section, and click the **Summary** link. A page that provides a compact view of the section assets is displayed.

## Add announcements to an application

Announcements appear on the **application contribution** page and provide general instructions and other application details for your contributors. An application can contain multiple announcements. When multiple announcements are present, they display in reverse chronological order, with the most recent announcement appearing first.

### Before you begin

You can add announcements to an application after you have defined an application and assigned co-owners.

### Procedure

1. On the application page, click the **Announcements** tab to create an announcement for the application.
2. Click **Create**, then enter an announcement for the application.

   Optional: Use the **Upload files** option to add one or more files to the announcement. Any attachment you add here is available to users as a link on the announcement.

## Search and filter in applications

You can search and filter on application sections in edit mode.

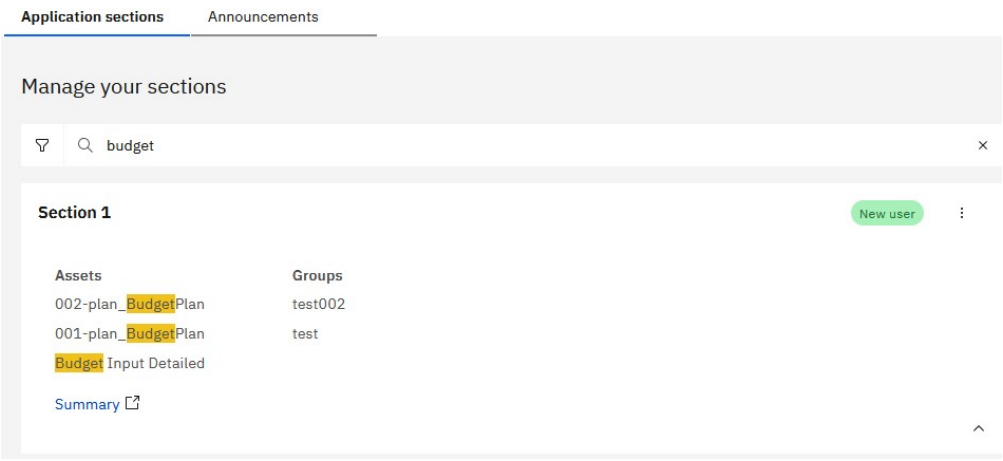
The search input field and filter button  are shown before section tiles.

Sections can be searched and filtered by assets, tags, attachments, and contributors.

### Procedure

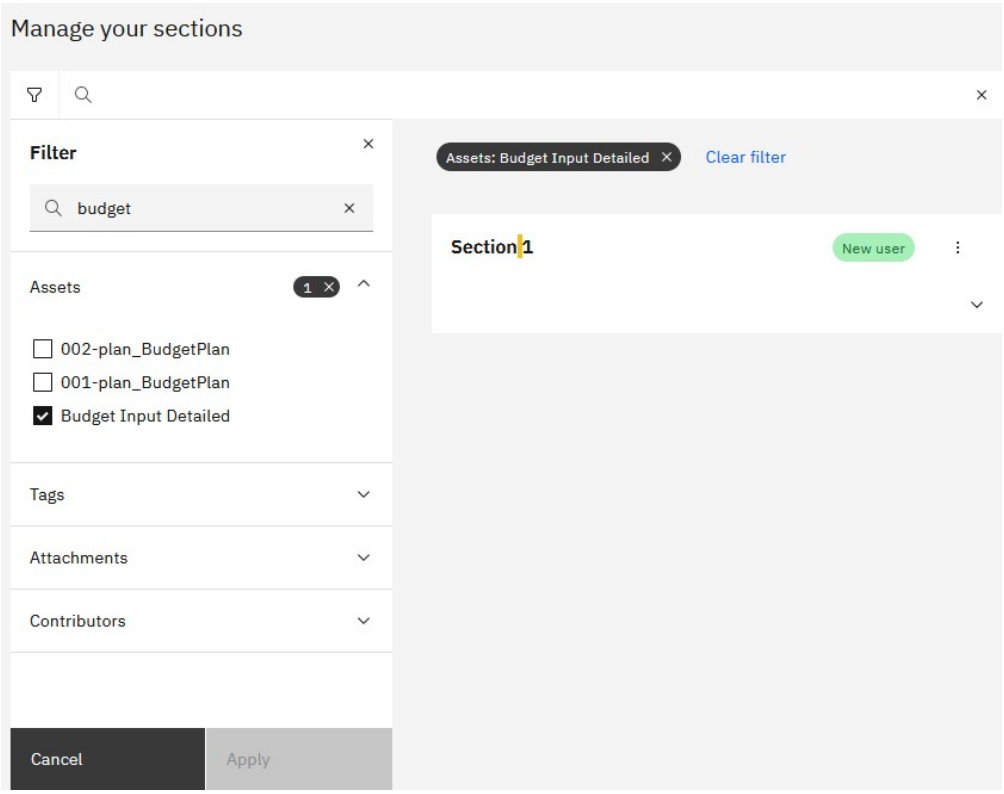
1. Open an application in edit mode.
2. On the **Application sections** tab, search or filter on sections.
   - To search for assets by name, in the search field type your keyword.

     The section tiles that contain the searched items are expanded, and the search keywords are highlighted.



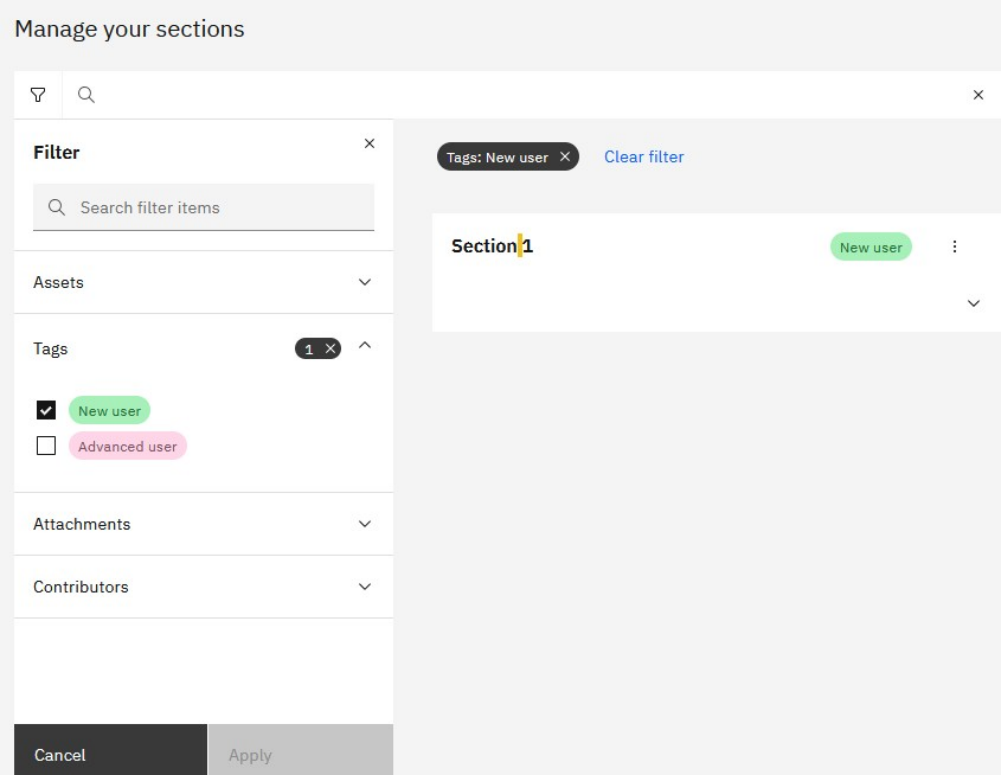

- To filter on sections, click the filter button to open the **Filter** panel, and use one of the following methods to filter on items:
  - Type a keyword in the search field.

    The search results appear below the search field. Click the result that you want to filter on. The items that contain the search result are expanded in the **Filter** panel. Select the related check box or button, and click **Apply**. The filter tag is added in the content panel, and the sections that contain the filtered results are shown.




  - Expand the items in the panel.

    Select the related check box or button, and click **Apply**. The filter tag is added in the content panel, and the sections that contain the filtered results are shown.




**Note:** Filter and search results are not saved when you reload a page.

# Manage an application

You can modify virtually any component of an application. You can add, remove, or reorder sections and assets in an application. You can also open or close the entire application, modify group assignments, and modify announcements.

You must have the required permissions to edit or manage an application. A user with the **Full control** permission becomes the application owner, and can grant permissions for this application to other users.

**Important:** When you perform any action that affects asset or group security for an application you must explicitly refresh security by clicking **Actions**, then **Refresh Permissions**. Actions that require an explicit security refresh include:

- Adding an asset to a section
- Removing an asset from a section
- Adding a user group to a section
- Removing a user group from a section
- Adding a co-owner with full access to an application
- Removing a co-owner with full access to an application

## Edit an application

1. From the Home menu ☰, click **Applications and Plans**.
2. On the **Applications and Plans** page, click the **Action** button ⋮ for the application you want to edit.
3. Click **Edit**.
   You can now:
   - Edit application details

Under **Application actions**, click **Edit**. You can edit application details such as name and description, upload an image, and add or remove co-owners.

- Open or close an application

  Click the **Application status** option to toggle the status of the application.

- Add a section

  Click **Add a section**, then define the section as described in "Define an application and assign co-owners" on page 682.

- Edit a section

  Click the **Edit section** icon on the section you want to modify. You can modify the section details such as the section name or description, add, remove, or reorder assets for a section, or add or remove contributors.

- Change the order of sections in an application

  Click the section handle and drag the section to a new position.

- Delete a section

  Click the **Delete section** icon for the section you want to remove from the application, then confirm the deletion.

- Modify announcements

  Click the **Announcements** tab to create or delete an announcement.

### Manage permissions for users in an application

You can manage permissions for an individual user after you create the application.

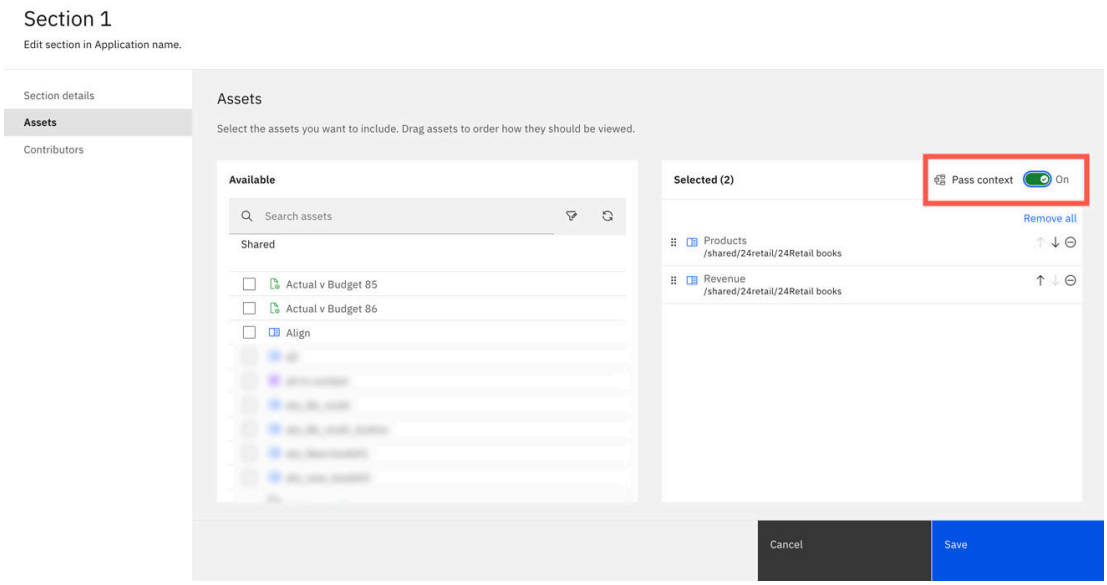
1. From the Home menu ☰, click **Applications and Plans**.
2. On the **Applications and Plans** page, click the **Action** button ⋮ for the application that you want to set permissions for.
3. Click **Set Permissions**.
4. In the **Permission** column, set the permission for an individual user. When you assign **Full control** permission to a user, the user is an application approver and has full control over the application.
5. Click **Save**.

## Pass context between books in an application

You can pass context between books that are attached as assets in an application that use the same synchronization group.

When you add books to sections in an application, enable the **Pass context** option to pass context between any books that have synchronization enabled and use the same synchronization group.

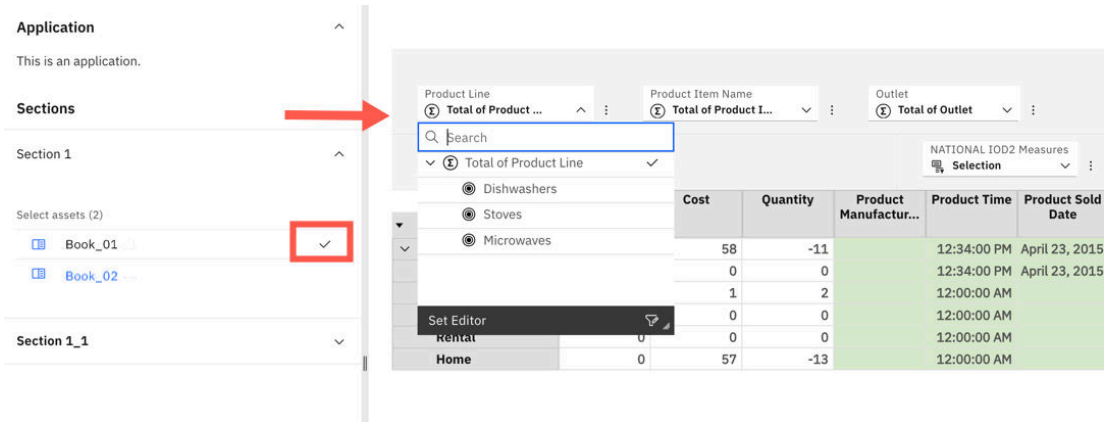
**Important:** Before you can pass context between books of the same application, you need to first setup synchronization for these assets in **Reports and Analysis**.



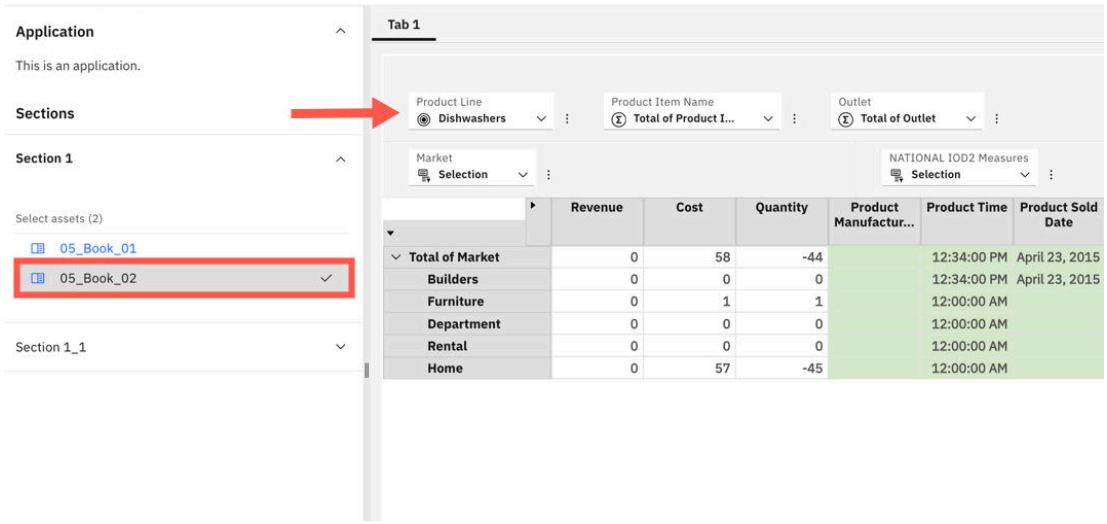

The **Pass context** feature supports all three synchronization groups (Book, Sheet, and Custom) as well as synchronization of hierarchies and gestures.

When you pass context between books that are synchronized at the book level, the source book is the one that opens first and has a check-mark next to its name in the application section. The target book is the book that you click on next.

In the following example, Book_01 and Book_02 are synchronized at the **Book** level. Book_01 is the source book in application `Section 1`.




When you change the **Product Line** to **Dishwashers** in Book_01 and go to Book_02, the hierarchy in Book_02 automatically updates to **Dishwashers**.




In books that are synchronized at the sheet level, you can choose the source tab but you cannot choose the target tab. The target tab is always the first tab of whichever book you click on after making changes to the source book.

Similarly, in custom group level synchronization, widgets that belong to the same group will be synchronized together when you enable **Pass context** for assets in an application. As members of a hierarchy that display in one widget in the source book change, the same members display in synchronized widgets that use the same hierarchy in the target book.

# Convert an application to a plan

When you convert an application to a plan, you can organize your assets in tasks, assign due dates, and require contributor submissions.

## About this task

You must have the **Full control** permission on an application to convert it to a plan.

When you convert an application to a plan, the following actions happen:

- The new plan is created with a Closed status.
- The application logo is retained as the plan logo.
- All application announcements are removed.
- All application sections are converted to plan tasks, and the status for each task is Closed.
- The section names from the application are retained and applied to the tasks in the plan.
- Any section guidance is retained and applied to the tasks in the plan.
- The assets and user groups from the application sections are applied to the tasks in the plan.
- There are no due dates set for any plan tasks.
- The **Require submission** option is set to **No** for all plan tasks.

## Procedure

1. Open the application management view. There are two ways to open an application for management:
   a) Click the application tile on the Planning Analytics Workspace Home page.
   b) Click the application name in the **Applications and Plans** page.
2. Click **Application actions**, then **Save as**.
3. Enter a name for the plan, then select the **Plan** option and click **OK**.

### Results

The plan opens in the plan management page. See “Manage a plan” on page 706 for details on modifying the plan.

## Open and use an application

Any user that is a member of a user group that is assigned to at least one application asset can open and contribute to a plan.

### About this task

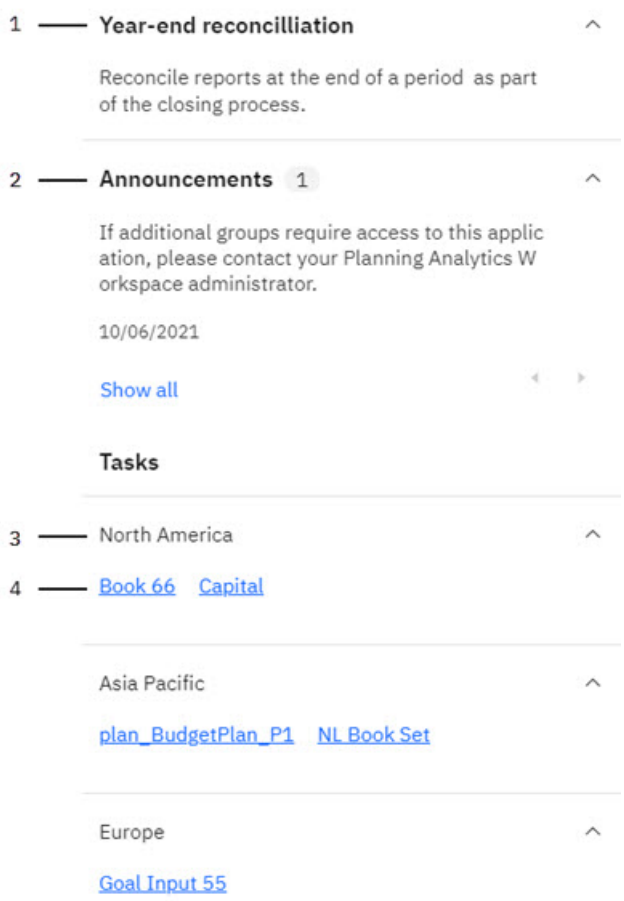
When you open an application, you can see and open only the sections and assets that have been assigned to a user group that you belong to. If an application contains five sections, but your user group has been assigned to three section, you'll see only those three sections.

### Procedure

1. There are two ways to open an application for contribution:
   a) On the Home page, click the **Applications and plans** tile, then click the **Actions** icon ⋮ on the application you want to use and select **Open**.
   b) On the Home page, click the **Home** icon ☰, then click **Applications and plans**. Click the **Actions** icon ⋮ on the application you want to use and select **Open**.

   The application opens to the contribution page. The first asset from the first section to which you have access is open and ready for your contribution.

   The left pane of the contribution page shows the sections and other information you need to work with the application.



**1 - Application name**

**2 - Application announcements**
The announcements display in reverse chronological order, with the most recent announcement appearing first. Click **Show all** to view all announcements or click the arrows to scroll through announcements.

**3 - Application sections**
You can see all sections that your user group has been assigned to.

**4 - Section assets**

2. Complete your contribution to the first asset.
3. Click the next asset you want to work on and complete your contribution. Repeat until you've completed all the contributions you want to make.

   You can work on sections or assets in any order.
4. When you're done, you can close the application from the Planning Analytics Workspace Home menu.

# Plans

A plan contains assets that are grouped in tasks. These tasks can represent discrete tasks or contributions that must be completed in a planning or budgeting process. Each task in a plan must be completed by contributing data to the books, views, and websheets in the task.

Plans are created by Planning Analytics Workspace administrators, analysts, and modelers. The plan creator can give owner permissions for the plan to other users. When you log in to Planning Analytics Workspace, the plans that you can use appear at the bottom of the home page on the **Your Applications** tab. Plans are identified by the icon.

You can view the details of a plan from either the home page or the **Applications and Plans** page. The details page shows the status, announcements, tasks, and assets for a plan.

While the details page is largely a 'read-only' snapshot of a plan, you can click any asset to open the full contribution page for the plan.

To view the details of a plan, use one of the following methods:

- If you are a user with the **Full control** permission, click the **Options** icon on the plan tile, then click **Details**. You can also click the **Options** icon for the plan in the **Applications and Plans** page, then click **Details**.
- If you are a contributor, click the plan tile on the home page to open the plan details. You can also click the plan name in the **Applications and Plans** page to view plan details.

## Create a plan

Create a plan to organize the books, views, and websheets used in your business processes for easier collaboration, and to provide guidance to process contributors.

Plans contain assets that are grouped into tasks which must be completed in a planning or budgeting process. You can assign contributors to a task, set due dates, and also define dependencies. Assigned contributors can complete the task by submitting data to the books, views, and websheets in the task.

You must be a Planning Analytics Workspace administrator, analyst, or modeler to create a plan. However, when you create a plan you can assign one or more co-owners to the plan. A co-owner has full control over the plan.

Any asset you want to include in a plan must reside in the Planning Analytics Workspace Shared folder.

There are three parts to plan creation:

1. Define a plan and assign co-owners
2. Add tasks with contributors and approvers
3. Add announcements

## Define a plan and assign co-owners

Create a plan to organize the books, views, and websheets used in your business processes and to provide guidance to process contributors.

### Procedure

1. On the Planning Analytics Workspace Home page, click **Applications and Plans**.
2. On the **Applications and Plans** page, click **Create**, then click **Plan**.

   You can also create a plan directly from the Home menu ☰. Click the **Home** menu, then click **New** and select **Plan**.
3. On the **Create a plan** page, under **Plan details**, enter a name and description for your plan. The description you provide is visible on the plan tile on the Planning Analytics Workspace Home page.

   Optional: Use the **Upload files** option to add one or more files to the plan overview. Any attachment you add here is available to users as a link immediately following the plan name and description.
4. Click **Image** to add an image to your plan. You can upload an image by clicking in the image box and selecting an image for the plan or dragging an image onto the image box.
5. Under **Co-owners**, select users that will be co-owners on this plan. Co-owners have full control over the plan and can modify all aspects of the plan.
6. Click **Create**.

   If you have more than one plan to create, you can duplicate the plan that you just created. On the plan page, click **Plan actions** and select **Save as**. Rename the duplicate and click **OK**. All details such as the **Assets**, **Contributors**, and attachments are copied over to the duplicated item.

### What to do next

You can now add tasks and announcements to your plan.

You can also make the plan available to other users by switching the **Plan status** toggle to **Open**.

## Specify data accountability in a plan

Data accountability allows plan owners to assign responsibility for specific slices of data in a TM1 model to Planning Analytics Workspace groups. This feature provides a structured workflow experience for submitting, reviewing, and approving data changes, similar to the TM1 Applications service.

With data accountability, you can perform the following activities:

- Assign members from a TM1 dimension to Planning Analytics Workspace contributor and reviewer groups.
- Track the workflow state of each assigned member.
- Submit, approve, or reject changes at both leaf and consolidated levels.
- View task history and receive notifications for key events.

Data accountability supports alternate hierarchies and subsets, allowing flexible member selection for accountability.

### Before you begin

Data accountability is not available by default. Enable data accountability from the Planning Analytics Administration page:

1. Go to the **Planning Analytics Administration** page.
2. Click **Features and Configuration**, then select the **Features** tab.
3. Click the **Data accountability** toggle until the Status is **On**.

To start data accountability in a TM1 database, the plan owner must have the **Admin** role in that database. Otherwise, a co-owner with **Admin** privileges must start the data accountability feature.

## Procedure

1. On the plan page, go to the **Database settings** tab.
2. Select a database from the **Database** dropdown list.
3. Click **Enable database** if the button is available.

   If the **Enable database** button is not available, the database is already available to use.
4. In the **Setup accountability** section, select the dimension and hierarchy for data accountability.

   You can only select hierarchies that have a single parent or root element. If you select a hierarchy that has multiple parents or root elements, an error message appears.
5. Select a subset of members to make available for data accountability tasks.
6. Click **Create**.

## Results

Each member in the data accountability hierarchy is treated as a node with a workflow state and is tracked in a cube in the TM1 database. The following workflow states, which are represented by numbers 0 to 4, are tracked in the cube:

*Table 7. Workflow states*

| Number | State | Description |
|---|---|---|
| 0 | Available | The node is available for a user to take ownership. |
| 1 | In progress | At least one item is in the `Available` state and at least one other item is in the `Reserved`, `Pending`, or `Complete` state.<br>The `In progress` state only applies to consolidated members. Leaf members cannot be `In progress`. |
| 2 | Reserved | A user has taken ownership of the node and data changes can be submitted for review. Ownership of a node must be released before a different user can take ownership. |
| 3 | Pending | The submission is awaiting approval or rejection. If a node is rejected, it is returned to the previous user that took ownership of the node. |
| 4 | Complete | The node and all descendants are approved. All nodes below a node in `Complete` state must also be `Complete`. No data changes are expected once the node state is `Complete`. |

State changes are triggered by the user actions such as the **Assign to me**, **Submit**, **Approve**, and **Reject** buttons in the plan. These actions also update ancestor and descendant nodes accordingly.

A full task history of all node interactions is available to plan owners. Contributors and reviewers have access to the task history for nodes that they can access. Users receive notifications for key events. All contributors receive notifications of task rejection, task approval, take ownership, and release ownership events. Reviewers and plan owners receive notification of revert task approval events.

Using data accountability will generate the following objects in the TM1 database:

- TurboIntegrator processes prefixed with `}Plans_` (do not modify or delete).
- A `}Plans_Everyone` group with a rule in the `}ClientGroups` cube to include all users.
- A `}Plans` dimension and cube to track plan metadata.
- A state cube for each Data Accountability task, including:
  - A generated dimension for the selected hierarchy.
  - A task dimension for each Data Accountability task.
  - The shared `}Plans_State` dimension for group assignments and node states.

## Add tasks to a plan and assign contributors and approvers

Tasks represent discrete steps or contributions that must be completed in a planning or budgeting cycle. When you add tasks to your plan, you can assign these tasks to contributors and approvers. You can also control the order in which tasks are completed by creating dependencies between the tasks.

### Before you begin

You can add tasks to a plan after you have defined a plan and assigned co-owners. You can add data accountability tasks to a plan after you specified data accountability in the plan.

### Procedure

1. On the plan page, go to the **Plan tasks** tab and click **Add task**.
2. Select the type of task that you are creating.
   - To add a regular task, click **Task**.
   - If you specified data accountability in your plan, click **Data accountability task**.
3. Under **Task details**, enter the task name and description.
4. Select one or more dependencies for the task. Dependencies let you automate your plan workflow by opening a task only when any dependent tasks are complete.

   The **Task dependency** option is available only when a plan contains two or more tasks. The option is not available when you create the first task in a plan. If you select a single dependency, the task automatically opens when the single dependent task is complete. If you select multiple dependencies, the task opens only when all dependent tasks are complete.
5. Enter a start and due date for the task.

   The due date indicates the deadline for contributions to the task, but it does not have any impact on the state of an open task. When the due date is reached or surpassed, an open task remains open until it is closed by an administrator.
6. Indicate whether the task requires a recorded submission.

   When a task requires submission, at least one user from one of the assigned contributor user groups must click **Submit** on the plan contribution page to confirm that work was completed for the task. A member of the approver user group then reviews the submission and either accepts or rejects it. If a task does not require a submission, the user can click **Complete** to indicate that a task is complete.
7. Under **Tags**, follow these steps to add one or more color-coded tags:
   a) Click into the **Tags** field, and click **Create new tag**

b) In the **Name** field, enter a label for your tag.

c) Choose a color for your tag.

Select from the colors that are available in the dialog box, or click **Custom color picker** and select a custom color. The color picker is different in different browsers. You can see what the tag will look like in the **Preview** field.

d) Click **Create**.

e) Expand the **Tags** field, select the tag that you just created, and click **Save** to add it to the task.

Any tags that you create are saved to the plan. This means that all tasks in a plan can share the available tags. Tasks can't access tags from other plans.

f) Click the **Edit a tag** icon next to a tag to edit or delete the tag.

**Important:** When you delete a tag, the tag is removed from all tasks in the plan that use the tag.

8. Use **Upload files** to add one or more files to the task. Any attachment that you add here is available to users as a link in the task.
9. Under **Assets**, select the assets (books, views, or websheets) from the **Available** section that you want to include in the task.

   Assets then display in the **Selected** list in the order that you select them. You can click and drag assets to different positions to change the ordering.

   **Tip:** In Planning Analytics Workspace 2.0.94 and later versions, you can select a tab to open by default in book assets.
10. Under **Contributors**, select the user groups that you want to assign as contributors to the task.

    You can select any combination of existing user groups or the Everyone virtual user group. The Everyone group assigns all users in your environment as contributors for a task.

    When a user group is assigned as contributors to a task, the security for the section assets is updated to provide the necessary access for the user group.

    **Important:** Plan security is applied only to Planning Analytics Workspace user groups. There are no links or dependencies to TM1 object security. It is the responsibility of the Planning Analytics Workspace administrator to ensure that the Planning Analytics Workspace user groups have the requisite TM1 security permissions to view and edit assets.
11. Under **Approvers**, select the user groups that you want to assign as approvers for each contributor group. Approvers review submissions and can either approve or reject them.
12. Click **Add task**.
13. Repeat these steps if you want to create additional tasks.

    You can also click **Duplicate task** to duplicate a task that you created to quickly create similar tasks.

    When you duplicate a task, most of the details such as the dependencies, contributors, and approvers are copied from the original task to the duplicated task. The following information in the duplicate task differs from the original task:

    - Task name - The duplicated task name includes the original name and appends an underscore and number at the end of the task name.
    - Start and due dates - Dates that occur in the past are reset. For the start and due dates to be copied over to the duplicated task, the start date must occur in the future.
    - Submission logs - Contributions and approvals are not copied over and submission logs are reset in the duplicated task.
    - Task status - The task status for duplicated tasks is set to **Closed**.

## What to do next

All plan tasks display under **Plan tasks** in the order that you create them. To change the order of tasks in a plan, click a task handle ⠿ and drag the task to a new location on the **Plan tasks** tab.

You can also add announcements to your plan.

To view the task summary page, on the **Tasks** tab, expand the task, and click the **Summary** link. A page that provides a compact view of the task assets is displayed.

## Add announcements to a plan

Announcements appear on the plan contribution page and can be used to provide general instructions and other plan details for your contributors. A plan can contain multiple announcements. When multiple announcements are present, they display in reverse chronological order, with the most recent announcement appearing first.

### Before you begin

You can add tasks to a plan after you have defined a plan and assigned co-owners.

### Procedure

1. On the plan page, go to the **Announcements** tab to create an announcement for the plan.
2. Click **Create**, then enter an announcement for the application.

   Optional: Use the **Upload files** option to add one or more files to the announcement. Any attachment you add here is available to users as a link on the announcement.
3. Click **Create**.

## Search and filter in plans

You can search and filter on plan tasks in edit mode.

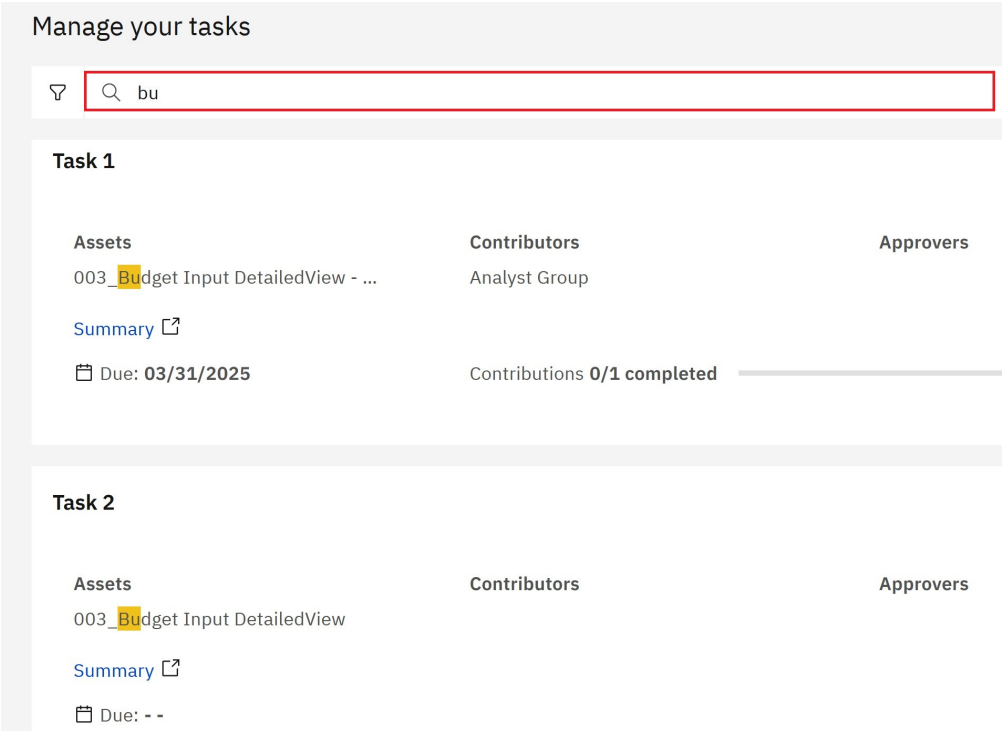
The search input field and filter button  are shown before task tiles.

Tasks can be searched and filtered by assets, tags, attachments, contributors, approvers, task dependencies, status, and task progress.

### Procedure

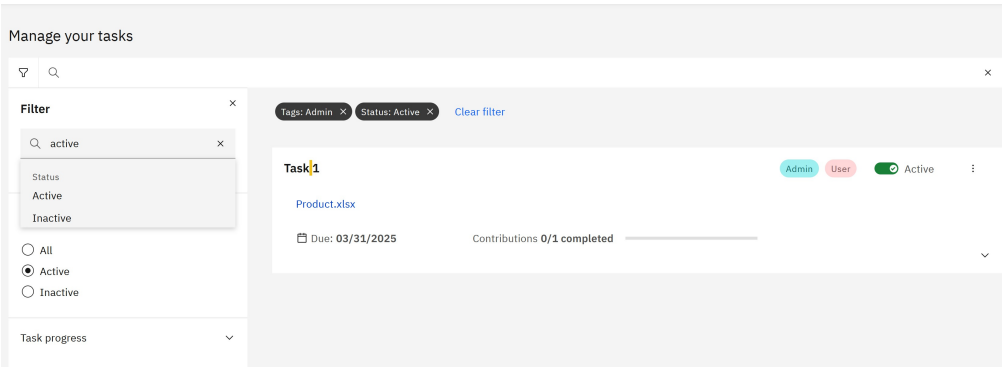
1. Open a plan in edit mode.
2. On the **Tasks** tab, search or filter on tasks.
   - To search for assets by name, in the search field type your keyword.

     The task tiles that contain the searched items are expanded, and the search keywords are highlighted.



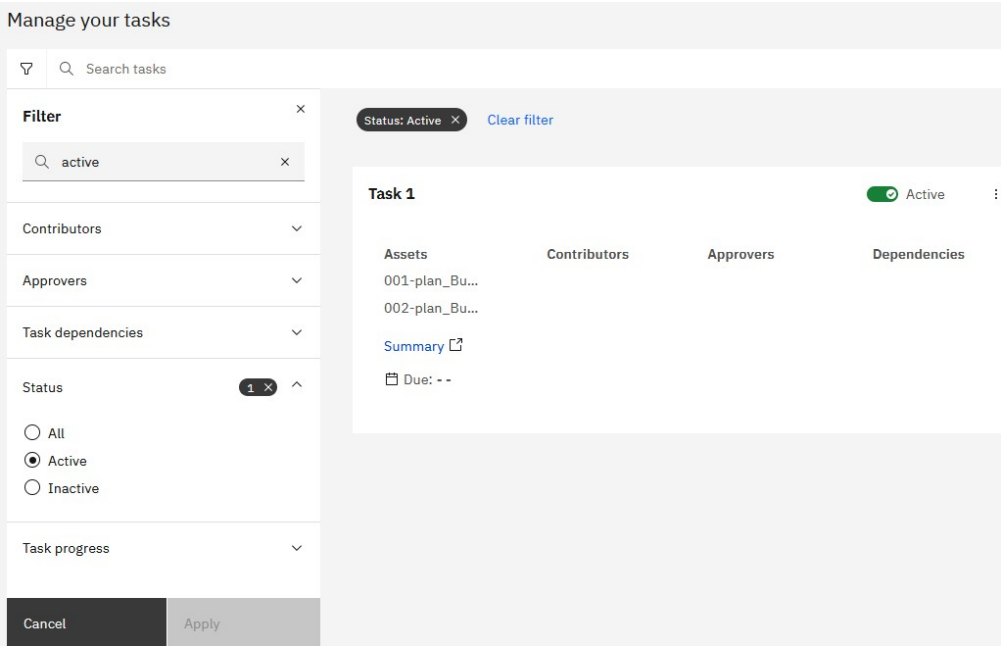

- To filter on tasks, click the filter button to open the **Filter** panel, and use one of the following methods to filter on items:
  - Type a keyword in the search field.

    The search results appear below the search field. Click the result that you want to filter on. The items that contain the search result are expanded in the **Filter** panel. Select the related check box or button, and click **Apply**. The filter tag is added in the content panel, and the tasks that contain the filtered results are shown.




  - Expand the items in the panel.

    Select the related check box or button, and click **Apply**. The filter tag is added in the content panel, and the tasks that contain the filtered results are shown.




**Note:** Filter and search results are not saved when you reload a page. The results are saved only when you refresh a plan.

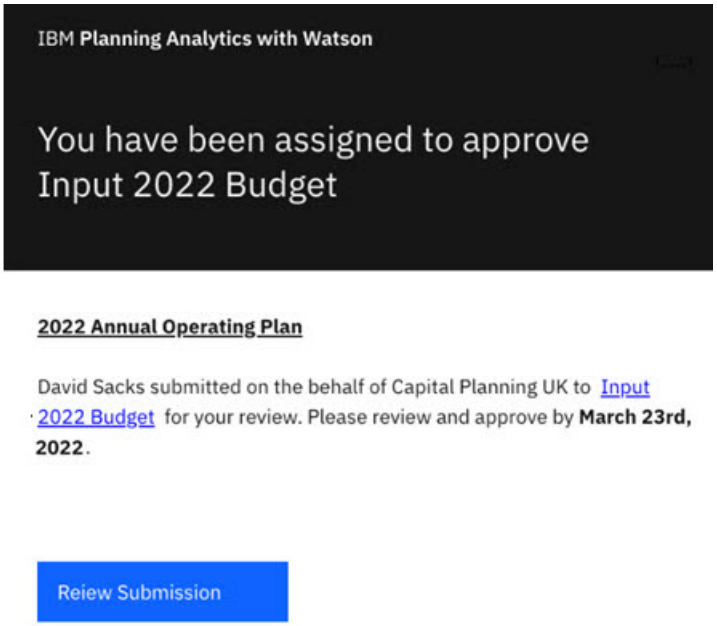
## Email notification in plans

To facilitate efficiency and workflow, an email is sent to relevant plan users when an action occurs in a plan.

The email includes details on the action and, when applicable, a link to the location in the plan where additional action can be taken or more information can be reviewed.

| Event | Email recipients |
|---|---|
| Task activation | Members of all groups that are assigned to the task receive a notification when a task is set to active. |
| Task submission | Plan owners and approvers that are assigned to the group that has submitted the task receive a notification when a task is submitted. |
| Task rejection | Members of all groups that submitted the task receive a notification when the task is rejected. |
| Task approval | Plan owners and the next in line approver receive an email notification when a task is approved. |
| Reverting a task approval | Plan owners and approvers receive a notification when a task approval is reverted. |
| Adding a co-owner to a plan | The newly added co-owner receives a notification. |
| Adding a contributor group to a task | All contributor members of the newly added group receive a notification. |
| Adding an announcement to a plan | Contributor groups assigned to active plan tasks receive a notification. |

For example, when a plan task is submitted, approvers who have been assigned to the group that submitted the task receive an email similar to this. The email would indicate the actual *task name*, *task contributor name*, and *group name* for the submission. The approver can click the **Review submission** button in the email to open the submission and either approve or reject it.




Additionally, you can send an email reminder to plan contributors from the **All tasks and submissions progress** section on the Overview tab.

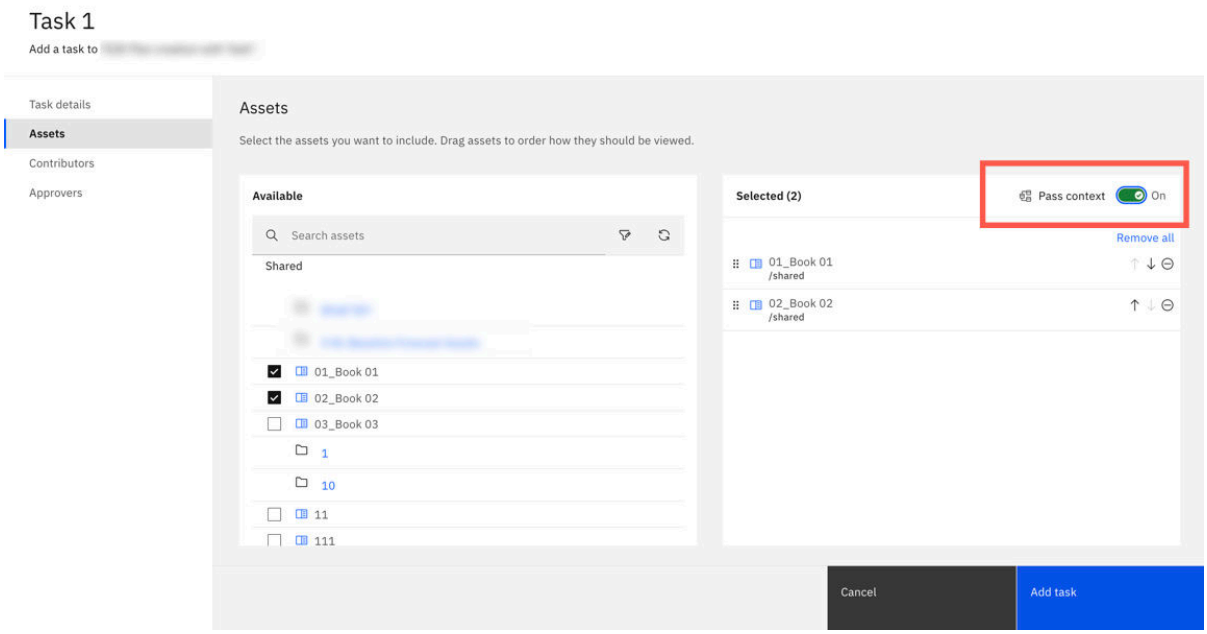
When an individual contribution on the **All tasks and submissions progress** panel is in the **Pending submission** or **Pending approval** state, an email icon is active at the end of the contribution record. Click the email icon to send a reminder that action is required to members of the group identified in the **Action pending** column.

## Pass context between books in a plan

You can pass context between books that are attached as assets in a plan that use the same synchronization group.

When you add books to tasks in a plan, enable the new **Pass context** option to pass context between any books that have synchronization enabled and use the same synchronization group.

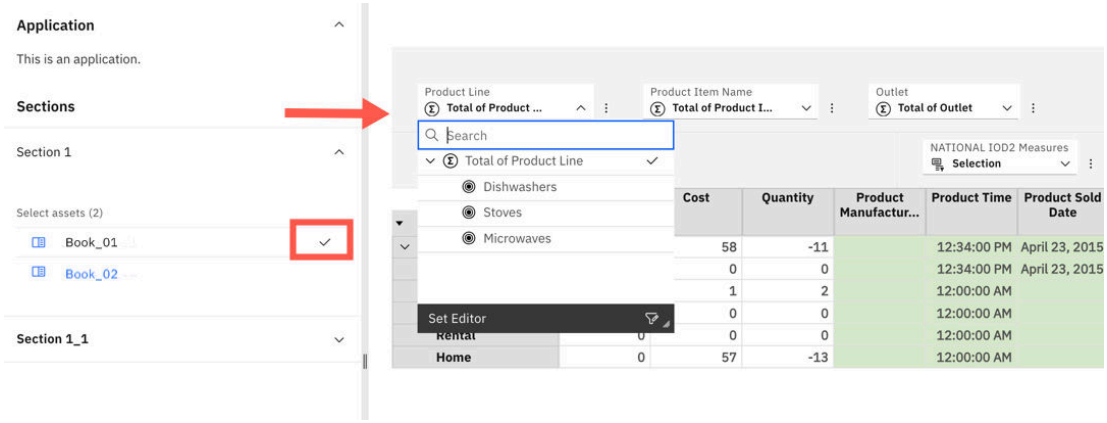
**Important:** Before you can pass context between books of the same plan, you need to first setup synchronization in these assets in **Reports and Analysis**.



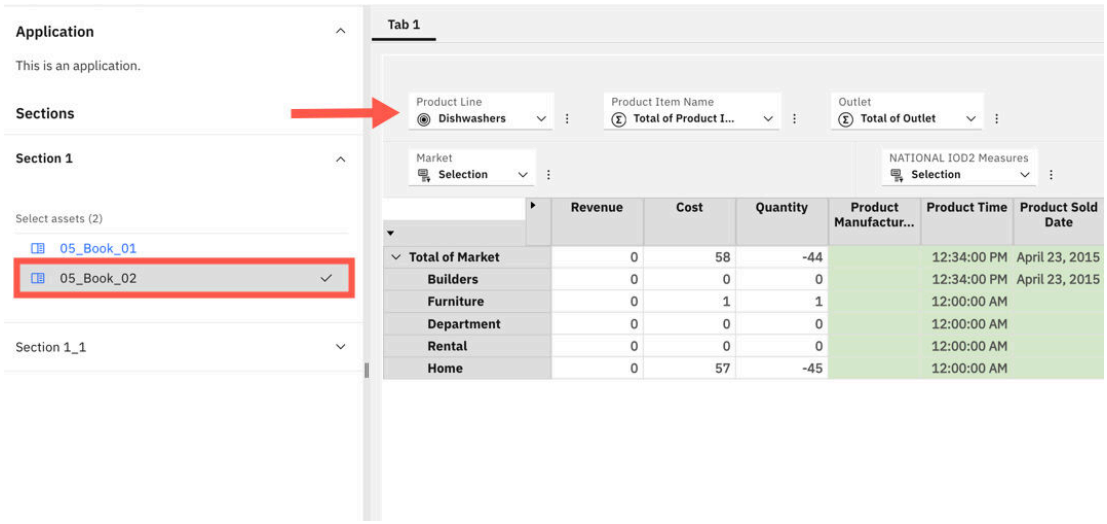

The **Pass context** feature supports all three synchronization groups (Book, Sheet, and Custom) as well as synchronization of hierarchies and gestures.

When you pass context between books that are synchronized at the book level, the source book is the one that opens first and has a check-mark next to its name in the plan task. The target book is the book that you click on next.

In the following example, Book_01 and Book_02 are synchronized at the **Book** level. Book_01 is the source book in application Section 1.




When you change the **Product Line** to **Dishwashers** in Book_01 and go to Book_02, the hierarchy in Book_02 automatically updates to **Dishwashers**.



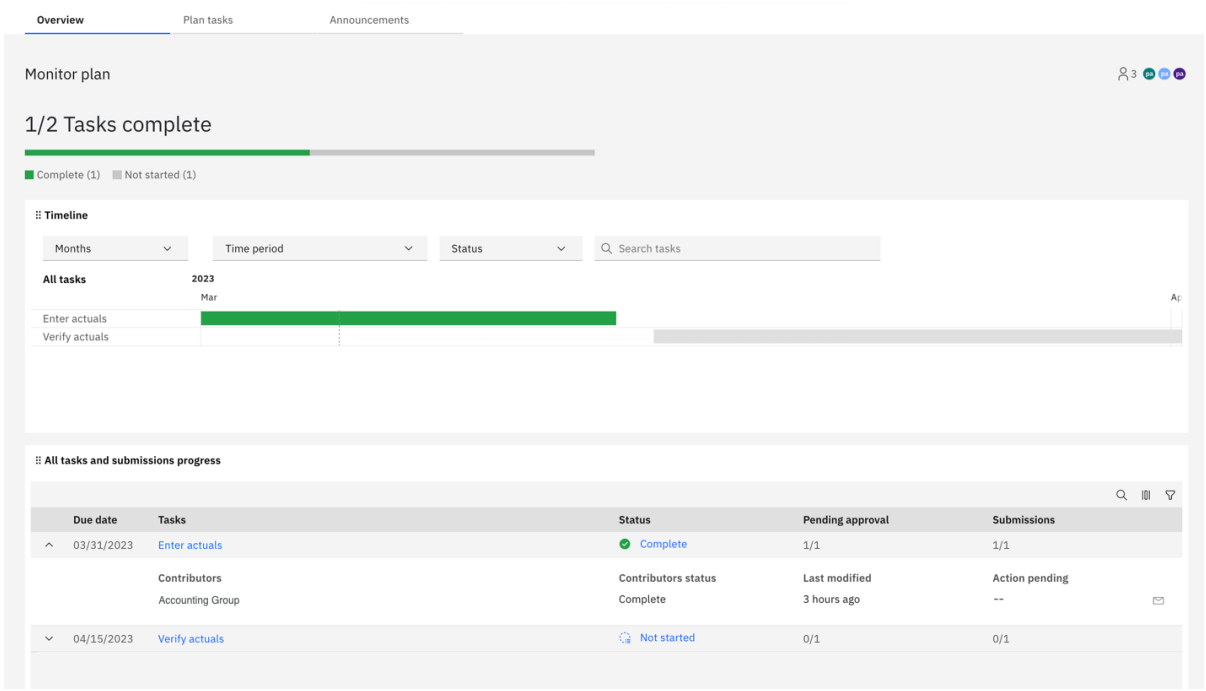

In books that are synchronized at the sheet level, you can choose the source tab but you cannot choose the target tab. The target tab is always the first tab of whichever book you click on after making changes to the source book.

Similarly, in custom group level synchronization, widgets that belong to the same group will be synchronized together when you enable **Pass context** for assets in a plan. As members of a hierarchy that display in one widget in the source book change, the same members display in synchronized widgets that use the same hierarchy in the target book.

## Track and report on plan progress with the plan Overview report

You can view a plan-level overview report to help you assess the progress of your plan.

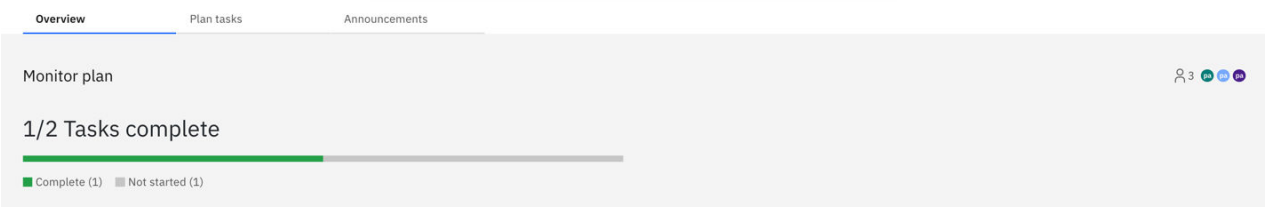
The report appears on the **Overview** tab when you are viewing plan details.




The overview report consists of a progress bar and two panels: **Timeline** and **All tasks and submissions progress**. You can click the handle next to the panel name to drag a panel to a different location on the Overview report.

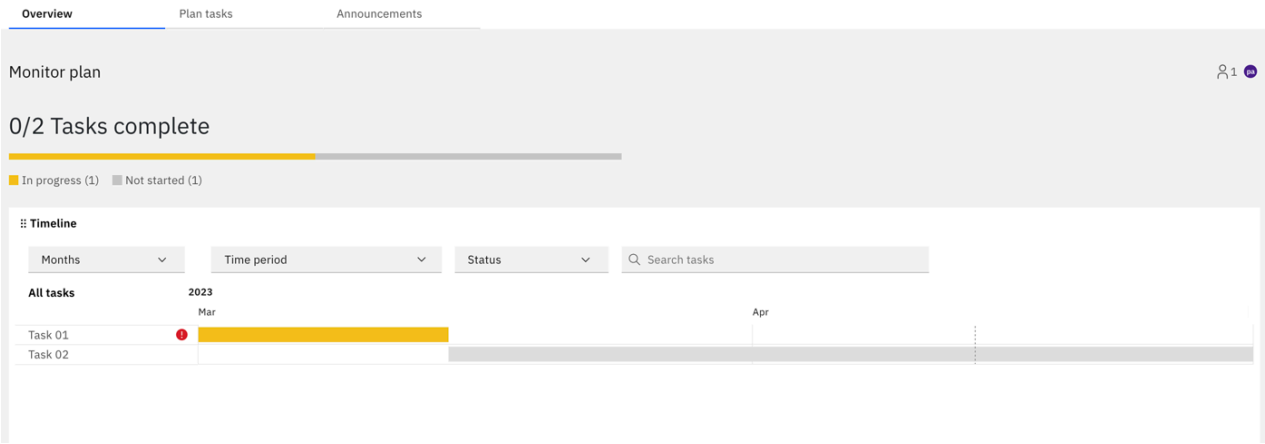
## Progress bar

The progress bar displays the progress of plan. You can see the number of tasks that were completed and the number of tasks that have not yet started.




## Timeline panel

The **Timeline** panel shows the status of tasks within a plan against a timeline, which can be configured to show either months or weeks.




If a task has only a start date or only an end date defined, the date is represented by a diamond on the timeline. Hover the pointer over the diamond to view the exact start or end date.

If a task has both a start date and an end date defined, a bar represents the date range on the timeline. Hover the pointer over the bar to view the start and end date of the task.

Any task that is not complete and overdue is identified by an urgent notice icon .

The **Timeline** panel includes advanced filtering. You can filter tasks by name, task status, and time period.

## All tasks and submissions progress panel

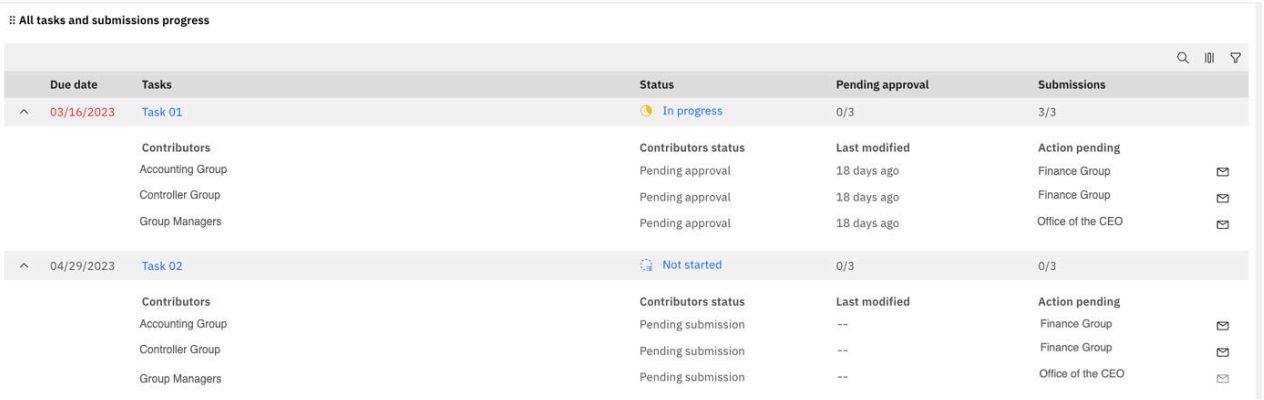
The **All tasks and submissions progress** panel displays progress for all submissions and individual tasks in a plan.

Expand each task to see status details for the individual submissions in that task.

Hover the pointer over a task name in the **Tasks** column to view the task description.

You can sort on any table heading and also search for tasks by name. You can customize the **All tasks and submissions progress** panel by choosing the columns that you want to see and filtering by task status.

When an individual submission is in the **Pending submission** or **Pending approval** state, an email icon is active at the end of the submission record. Click the email icon to send a reminder that action is required to members of the group that is identified in the **Action pending** column.




# Track tasks with the My tasks tab

The **My tasks** tab provides information about your current and completed tasks.

The **My tasks** tab is divided into three sections: **Task summary**, **To do**, and **Done**.

## Task summary

The **Task summary** section provides at-a-glance information about your pending and completed tasks, allows you to search for tasks with certain criteria, and allows you to change the layout of the tab.

This section has the following features:

- The search bar allows you to search the content in the **To Do** and **Done** sections by task name.
- The filter icon allows you to filter the tasks in the **To Do** and **Done** sections by the action required and the status of the task.
- The date range selectors MM/DD/YYYY MM/DD/YYYY allow you to filter tasks by their due date. You can search for tasks due within a range of time, and the tab only displays tasks in that period.
- The view switcher allows you to change the layout between table view and tile view.
- The settings button has different options depending on the type of view you choose. If you choose the table view, you can adjust the size of the rows, and in tile view you can show or hide descriptions and thumbnails.
- The refresh button refreshes the page.

## To do

The **To do** section has information on tasks requiring further action.

The **To do** section has the following information:

- Due date: when the task is due.
- Name: the name of the task. Clicking on the name of a task opens it. From here, you can contribute, approve, or reject the task.
- Action required: the type of action required to advance the task to the next step.
- Action pending by: the users required to complete the action in the Action required section.
- Description: a description of the task or required action.
- Comment: comments added about the task or required action.

## Done

The **Done** section has information on the tasks you completed.

The **Done** section has the following information:

- Due date: when the task is due.
- Name: the name of the task.
- Status: the current status of the task.
- Approval done by: the user who approved the task. If the task has not been approved, this section will state that the task has not been approved, or if approval is not required.
- Description: a description of the task.
- Comment: comments added about the task.

# Contribute to a plan

Any user that is a member of a user group that is assigned to at least one plan asset can open and contribute to a plan.

## About this task

When you open a plan for contribution, you can see and open only the tasks and assets that have been assigned to a user group that you belong to. If a plan contains five tasks, but your user group has been assigned to three tasks, you'll see only those three tasks.

## Procedure

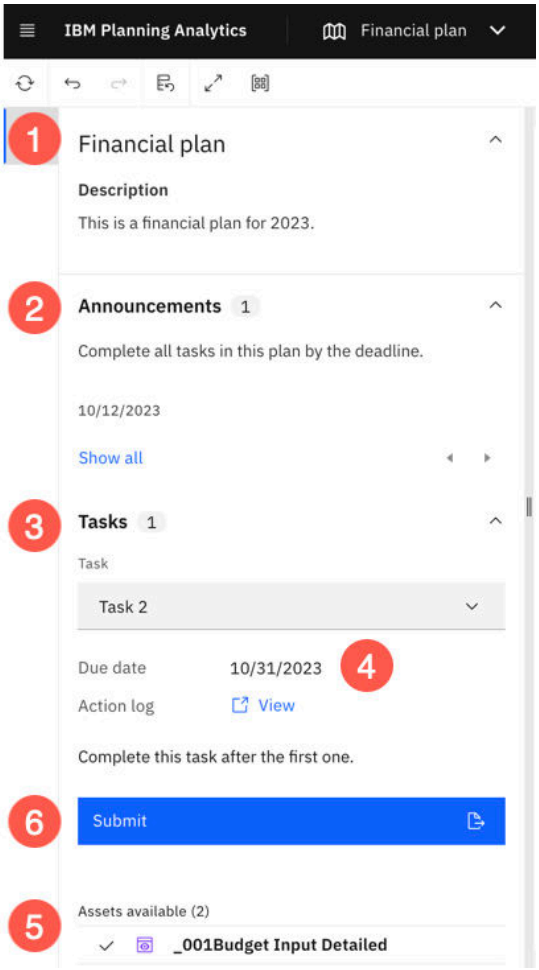
1. There are several ways to open a plan or task for contribution:
   a) On the Planning Analytics home page, click the **Applications and plans** tile, then click the **Actions** icon ⋮ on the plan you want to use and select **Open**.
   b) On the home page, click the **Home** icon ☰, then click **Applications and plans**. Click the **Actions** icon ⋮ on the plan you want to use and select **Open**.
   c) At the plan administrator's discretion, you might receive an email notification informing you when a task to which your group has been assigned is opened. The email contains a link directly to the task in the plan. You can click the link to open the task and begin your contribution.

   The plan opens in the contribution page. If you open the plan from the home page, the first asset from the first task to which you have access is open and ready for your contribution. If you open the plan from a link in an email notification, the specific task mentioned in the email is opened.

   The left pane of the contribution page shows the tasks and other information you need to work with the plan.

   **Note:** If you are assigned tasks for contribution and approval, the tasks are separated under these two tabs on the contribution panel: **Approve tasks** and **Contribute to tasks**

   If you close the contribution panel, the interface retains your preference and the contribution pane stays closed the next time you log in and open any plan or application unless you open the contribution panel again.




**1**

Plan name

**2**

Plan announcements. The announcements display in reverse chronological order, with the most recent announcement appearing first. Click **View all** to view all announcements or click the arrows to scroll through announcements.

**3**

Plan tasks. You can see all tasks that your user group has been assigned to in the **Task** list.

**4**

Select a task from the **Task** list to see the **Due date**.

**5**

Task assets

**6**

Submit button. Tasks in a plan can be configured by an administrator to require submission. If a task requires submission, the **Submit** button is available.

If a task does not require submission, click **Complete** to indicate that the task is complete.

2. Complete your contribution to the first asset.
3. Click the next asset you want to work on and complete your contribution. Repeat until you've completed all your required contributions.

   Though tasks are numbered sequentially, you can work on tasks or assets in any order.
4. If a task requires explicit submission, click the **Submit** button after you've finished working on all assets in the task.

If your task does not require submission, click **Complete** instead. Confirm completion by clicking **Complete** in the window that opens.

5. If the task requires submission and you are a member of multiple groups that are assigned to the task, click the **Select** list and choose the user group on whose behalf you are submitting your plan contributions. Optionally, you can choose **Submit all groups** to submit the plan on behalf of all groups to which you belong.

   If you are a member of only one user group assigned to the task, or if just a single user group is assigned to the task, you can't choose a user group; the submission is automatically made on behalf of your user group. You can also optionally enter a message to accompany your submission. The message can be viewed by your Planning Analytics Workspace administrator while managing submissions.
6. Click **Submit** to confirm the submission.

   When your submission is complete, a status message displays below the task assets indicating the time and date of your submission.
7. When you're done working, you can close the plan from the Planning Analytics Workspace Home menu.

# Manage a plan

You can modify virtually any component of a plan. You can add, remove, or change the order of tasks and assets in a plan. You can modify group assignments, due dates, and submission requirements. You can also open or close the entire plan or individual plan tasks. If your plan includes tasks that require submission, you can manage group submissions as well.

## About this task

You must have the required permissions to edit or manage a plan. A user with the **Full control** permission becomes the plan owner, and can grant permissions for this plan to other users.

**Important:** When you perform any action that affects asset or group security for a plan you must explicitly refresh security by clicking **Plan actions**, then **Refresh permissions**. Actions that require an explicit security refresh include:

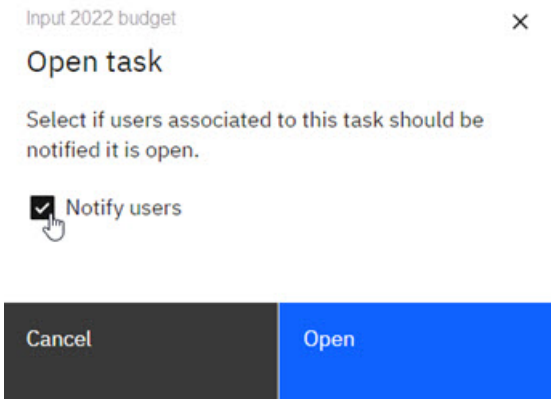
- Adding an asset to a task
- Removing an asset from a task
- Adding a user group to a task
- Removing a user group from a task
- Adding an approver with full access to a plan
- Removing an approver with full access to a plan

While managing a plan, it's possible that other users might contribute submissions or otherwise modify the plan. You can click **Plan actions**, then **Refresh plan** at any time to update the plan management page.

## Procedure

1. Open a plan for editing or management:
   a) From the Home menu ☰, click **Applications and Plans**.
   b) On the Applications and Plans page, click the **Action** button ⋮ for the plan you want to edit.
   c) Click **Edit**.
2. To open or close a plan, click the **Plan status** option to toggle the status of the plan.
3. To open or close an individual task, click the **Status** option to toggle the status of the task.

   When you open a task that includes at least one asset and has at least one user group assigned to it, you'll receive the option to **Notify users** when you open the task.




If you select the **Notify users** option and then click **Open** to open the task, all users in the groups assigned to the task receive an email notification indicating that their contribution to the plan is required. The email includes a direct link to the plan task.

If the Everyone group is assigned to the task, email is **not** sent to members of the Everyone group when the task is opened.

**Note:** The **Notify users** option appears only when you open a task in a plan that is open. If you open a task in a closed plan, there is no option to send email notification to users.

4. To edit a task, click the **Edit** icon on the task.
5. Use the **Task details** tab to modify the task name, description, due date, submission requirement, or status.
6. Use the **Assets** tab to add, remove, or reorder the assets for a task.
7. Use the **Groups** tab to add or remove groups for the task.
8. If a task requires submission, click **Submissions** tab to view a report of group submissions for the task.

   The **Submissions** page shows the date on which each group submitted their contributions, along with any message that the submitter provides.

   If the submission is not satisfactory, you can click **Reset** to rest the submission status for any group. The group must then contribute a revised submission. When you reset a submission, groups and users are not notified, but the submission status message for the group is removed from the plan contribution page and the status icons for the task are reset.
9. To add a task to a plan, click **Add task**, then define the task as described in "Define a plan and assign co-owners" on page 692.
10. To delete a task from a plan, click the **Delete** icon, then confirm the deletion.
11. To modify plan announcements, click the **Announcements** tab.
    a) Click **Create** to add an announcement.
    b) Select an existing announcement and click **Delete** to remove an announcement.
12. To add or remove approvers to a plan, click the **Action** button ⋮ for the plan you want to edit.
    a) Click **Set Permissions**.
    b) In the **Permissions** column, set the permission for an individual user. When you assign **Full control** permission to a user, the user is a plan approver and has full control over the plan.
    c) Click **Save**.

## Using the task list and task flow views

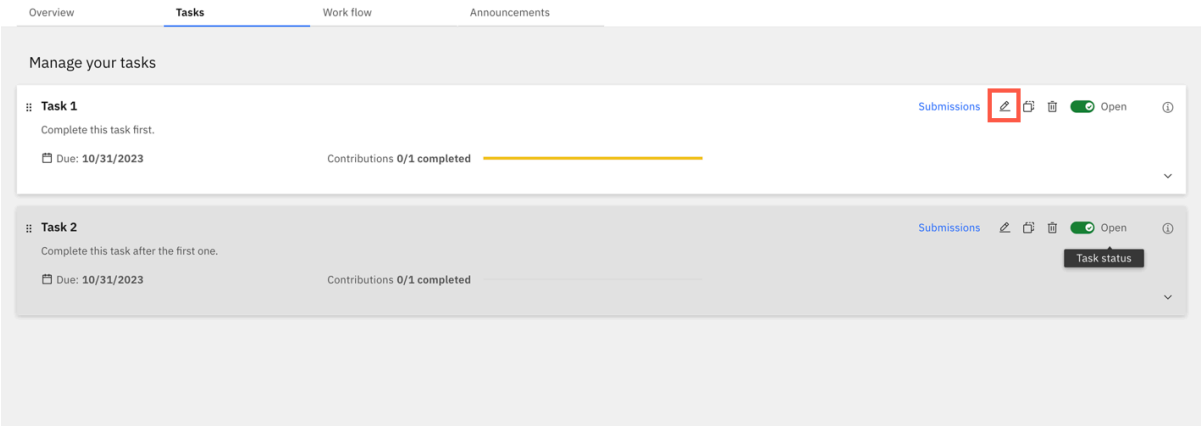
When a plan is in edit mode, you can create and manage tasks by using the task list view under the **Tasks** tab or the task flow view under the **Work flow** tab.

Changes in one view are reflected in the other view. However, not all task features are available in both views.

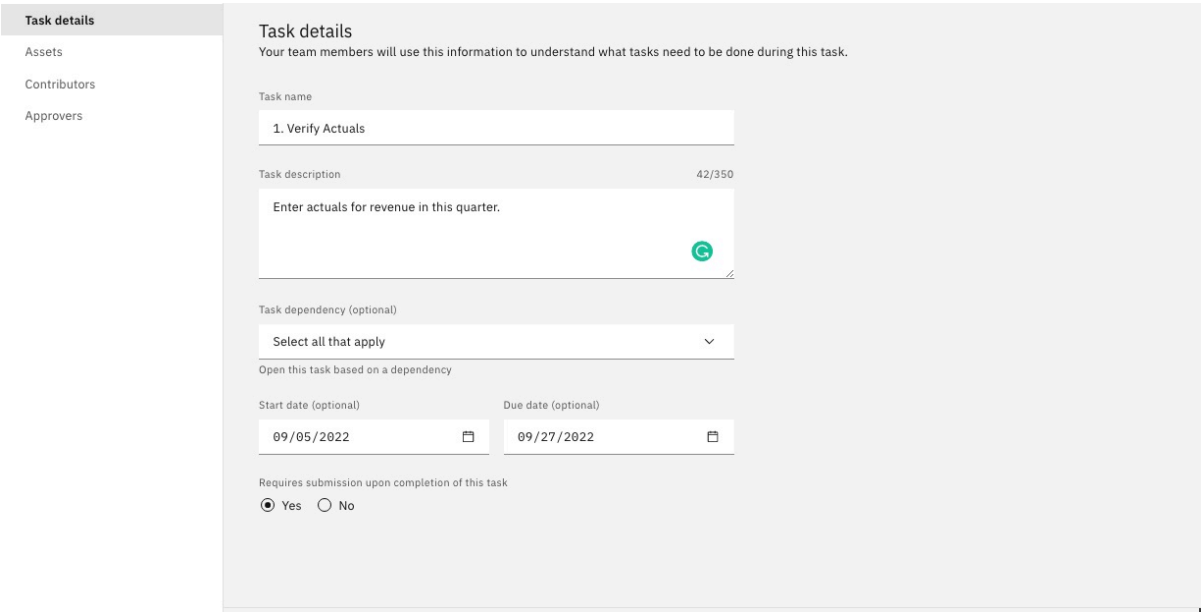
For related information, see “Define a plan and assign co-owners” on page 692 and “Manage a plan” on page 706.

## Task list view

The task list view provides the basic user interface for working with plan tasks. Here you can add and delete tasks, and open and close the tasks.




Click the edit task icon to specify task dependencies, assets, contributors, and approvers.

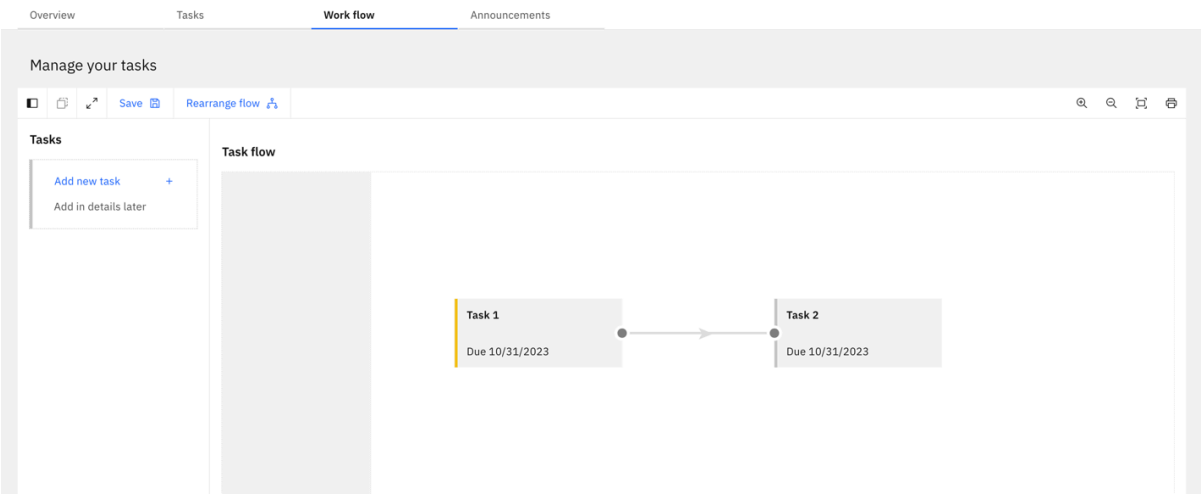



Changes in this view, such as task dependencies, are reflected in the task flow view under **Work flow**.

## Task flow view

The task flow view under the **Work flow** tab provides a more visual user interface to work with plan tasks.

In this view, you can create tasks and define task dependencies. When you create a task in this view, the task is shown under the **Tasks** pane. To define the task dependencies in this view, drag the tasks from the **Tasks** pane to the **Task flow** pane. Then, hover the cursor over the right side of the task, and draw a line toward the dependent task. A visual task flow diagram is created with lines that link the dependent tasks. You can rearrange the position of the tasks in the diagram. The diagram is saved when you save the plan.




Changes in this view are reflected in the task list view.

You can edit a task directly from the **Task flow** pane. Hover the cursor on a task, then click the **Edit** icon to open the task editor.

You can also use full screen mode to maximize the canvas when working in the **Task flow** pane. Click the Full screen icon to enter full screen mode, where the **Task flow** pane and the **Tasks** list occupy the entire screen.

Click the **Export** icon if you want to save the task flow as a PDF file.

## Manage plan submissions

Plan approvers can manage task submissions and can review logs that capture the history of submissions.

### About this task

Plan owners, co-owners, and designated approvers can accept or reject a plan task that has been submitted by a member of a group that is assigned to the task. They can also use the **Submissions** link on a task to open the **Submissions log**, where they can review the status of all tasks that require submission.

**Note:** Contributors can also open the **Submissions log**, but contributors can view the status only of the tasks to which they have been assigned.

When a contributor submits a plan task, an email notification is sent to the task approvers, as well as the plan owners and co-owners. The email includes a direct link to the plan task, where approvers can approve or reject the submission and provide commentary.

To take action on a submission, click the link in the email you receive or open the task in contribution mode.

**Note:** If you are assigned tasks for both contribution and approval, when you open a plan, your tasks for submission will display under the **Contribute to tasks** tab in the contribution panel and those that require approval, display on the **Approve tasks** tab.

When a task has been submitted, approvers can see the **Approve/Reject** button for he task, along with a history of submissions for the task.

### Procedure

1. Click **Approve/Reject** for the task you want to approve or reject.
2. Select the group submission for whom you want to approve or reject the submission.
3. Enter comments that describe your disposition on the submission and click **Reject** or **Approve**.

If you **Reject** the submission, the task remains open for contribution and the submitter is notified of the rejection. The contributor can then modify the assets and resubmit the task when ready.

If you **Approve** the submission, the task is closed for contribution and the next-in-line approver (if any) and the plan owners and co-owners are notified of the approval.

After you approve a submission, that task no longer displays in the contribution panel.

4. To review the Submission log for a task, click the **Submissions** link on the task in edit mode.

   The Submissions log includes three tabs.

   a) Click **Pending submission** to view all tasks requiring submission that have not been submitted.

   b) Click **Pending approval** to view all contributions that have been submitted but not yet approved or rejected.

   c) Click **Completed** to view all submissions that have been approved.

   You can reset task submissions that are under **Pending approval** and **Completed** by clicking the menu next to the submission and selecting **Reset submission**. To reset all submissions, click **Reset all**. All tasks are reset to an open state and any contributions or changes to assets are discarded.

   Plan owners and co-owners can also revert approvals for submissions that were previously approved. Submissions that are fully approved are under **Completed**. Submissions that require multiple approvals and are partially approved can be found under **Pending approval**. To revert an approval, click the menu next to the submission and select **Revert approval**. Select the approver from whom you want to remove approval.

   **Note:** Where there are multiple approvers for a task, when you revert an approval, approval is also removed from any next in line approvers.

## Convert a plan to an application

When you convert a plan to an application, you retain all the assets and groups from the plan, but any artifacts related to due dates and submission requirements are removed.

### About this task

You must have the **Full control** permission on a plan to convert it to an application.

When you convert a plan to an application, the following actions happen:

- The new application is created with a Closed status.
- The plan logo is retained as the application logo.
- All announcements from the plan are removed.
- All plan tasks are converted to application sections.
- The task names from the plan are retained and applied to the sections in the application.
- Any task guidance is retained and applied to the sections in the application.
- The assets and user groups from the plan tasks are applied to the section in the application.
- You cannot set due dates for application sections. All due dates from the plan are removed.
- You cannot require submission for application sections. All submission requirements from the plan are removed.

### Procedure

1. Open the plan management view. There are two ways to open a plan for management:
   a) Click the plan tile on the Planning Analytics Workspace Home page.
   b) Click the plan name on the **Applications and Plans** page.
2. Click **Plan actions**, then **Save as**.
3. Enter a name for the application, then select the **Application** option and click **OK**.

### Results

The new application opens in the application management page. See "Manage an application" on page 686 for details on modifying the application.

## Automate plan actions with workflow processes

Workflow processes allow plan owners to automate actions in a TM1 database when specific events occur in a Planning Analytics Workspace plan. These actions are triggered by TurboIntegrator processes and can be configured for both contributor and reviewer tasks.

You can configure TurboIntegrator processes to run automatically when the following events occur in a plan:

- A task is activated or deactivated.
- A contributor takes ownership of a task.
- A contributor submits a task.
- A reviewer approves or rejects a task.

The TurboIntegrator processes run under the credentials of the user who triggered the event. The user must have **Read** access to the process in the TM1 database.

The workflow processes can perform specific actions, including the following actions:

- Assigning security groups
- Locking or unlocking data slices by using security overlay
- Updating metadata or cube values based on plan state

The plan does not display an error message if the process fails to run. Workflow processes run after the plan state updates. Therefore, if a user takes ownership of a task, the state cube updates before the process runs.

### Parameters passed to the process

When an event occurs, the configured TurboIntegrator process runs in the TM1 database that is associated with the plan. The plan passes the following parameters to the process:

*Table 8. Parameters passed from the plan to the process*

| Event | Task | Data accountability task |
|---|---|---|
| Activate task | Plan ID, Task ID | Plan ID, Task ID |
| Deactivate task | Plan ID, Task ID | Plan ID, Task ID |
| Take ownership | N/A | Plan ID, Task ID, Member |
| Submit the task | Plan ID, Task ID | Plan ID, Task ID, Member |
| Approve the task | Plan ID, Task ID | Plan ID, Task ID, Member |
| Reject the task | Plan ID, Task ID | Plan ID, Task ID, Member |

- Plan ID and Task ID are unique identifiers that are generated when the plan and task are created.
- Member refers to the node in the data accountability hierarchy.

### Enabling workflow processes

Workflow processes are not available by default. Enable workflow processes from the Planning Analytics Administration page:

1. Go to the **Planning Analytics Administration** page.
2. Click **Features and Configuration**, then select the **Features** tab.

3. Click the **Workflow processes** toggle until the Status is **On**.

# Chapter 5. Explore scorecards

Scorecards reflect the strategic goals of an organization. Using scorecards, you can identify how well objectives are being met by comparing targets to actual results. Visual status indicators such as traffic lights, trend icons, and colors are used to help you to quickly evaluate performance.

In IBM Planning Analytics Workspace, you can add existing scorecards to your books, and analyze data by selecting different time periods, metrics, and dimensions. You can also create visualizations from scorecards, such as impact diagrams and strategy maps.

## Scorecards

A scorecard is a collection of performance metrics that are designed to reflect the strategic goals of your business unit or organization.

The information in a scorecard identifies how well the objectives are being met by comparing planned to actual results. Scorecards can also show information for the different organizations in your business. By using visual status indicators such as traffic light, and trend icons, scorecards can help users to quickly evaluate performance.

A scorecard combines data and dimensions into interactive diagrams and visualizations that you can share with other users. Scorecards include the following elements:

**Metrics**
A measure or key performance indicator (KPI) that conveys the performance of an important area of your business. Examples include Profit, Revenue, and Expenses.

**Metric indicator**
A measure of performance, status, or trend for a key area (metric) of your business. A metric indicator compares current results to target values. For example, Score, Status, and Trend.

### Impact diagram

Impact diagrams illustrate the positive and negative relationships between the metrics in your metrics cube. This type of diagram shows how the business works by displaying how one metric influences another.

An example for an impact diagram might show how Revenue and Expenses influence Profit, which then affects Bonuses and Research Funding.

Impact diagrams display traffic light and trend indicators that show the status and the trend of each metric in the diagram. You can filter for different contexts in the impact diagram. The traffic light and trend indicators update with new values for the selected dimension.

When you double-click a metric in an impact diagram, the selected metric become the focus metric. Views, visualizations, and other widgets on the current dashboard also update automatically to show data in context of the focus metric.

### Strategy map

A strategy map is a visualization that tracks business performance by perspectives, objectives, and metrics.

A strategy map shows the status of metrics with traffic lights and trend indicator icons. A strategy map organizes perspectives, objectives, and metrics into the following hierarchy:

- A strategy map can have multiple perspectives.
- Each perspective can have multiple objectives.
- Each objective can have multiple metrics.

© Copyright IBM Corp. 2009, 2025

The standard perspectives for a strategy map include the following items:

- Financial performance
- Customer knowledge
- Internal business processes
- Learning and growth

Connections in a strategy map display as directional arrows to show a visual relationship or flow between the objectives in the diagram.

When you double-click a metric in a strategy map, the selected metric become the focus metric. Views, visualizations, and other widgets on the current dashboard also update automatically to show data in context of the focus metric.

## Custom diagram

A custom diagram is a strategy map that has a custom image and shows metrics with dimensional context onto the image as data points.

These examples of custom diagrams are available:

**Geographical maps**
Show a regional focus of your organization.

**Process diagrams**
Shows metrics in the context of a process flow.

A custom diagram displays the metric and context dimension names with traffic light and trend indicator icons as an overlay or layer on the selected image.

# Metrics cubes

A metrics cube is a special type of cube that provides the basis for scorecard solutions and scorecard diagrams.

A metrics cube monitors multiple metrics and metric indicators. The primary feature of a metrics cube is that it shows the current relative status of many rows in a table. It displays the current trend of many measures simultaneously.

The standard scorecard layout for a metrics cube is as follows:

- Row title dimension: metric dimension
- Column title dimension: metric indicator dimension
- Context dimensions: time, geography, and other data context dimensions

A metrics cube combines a metric dimension and metric indicator dimension with your other regular dimensions. Metrics cubes have the same properties of other cubes.

A metrics cube must contain the following dimensions as a minimum:

- A metric dimension
- A metric indicator dimension
- A time dimension

### Metric dimension

The metric dimension contains your collection of important measures or key performance indicators (KPI) that you want to monitor in your business or organization.

These measures are called metrics and each identifies one aspect of performance, such as Gross profit, Revenue, or Product cost. You can monitor the actual performance of a metric and compare it to expected

or target values by combining it with metric indicators to provide the additional details about status, score, and trend.

### Metric indicator dimension

A metric indicator dimension provides more information about your key performance indicators (KPI) or metrics. Examples of metric indicators include Score, Status, and Trend.

The metric indicators measure the performance, status, and trends in key areas of a business by comparing current results to target values. For example, the Actual, Target, and Tolerance indicators for a metric are typically used to calculate the related Score, Status, and Trend indicators.

## Traffic light status indicator

A traffic light status indicator shows if metrics are meeting set targets.

The status is indicated by the color and the shape of the icon.

*Table 9. Metric indicator traffic light status icons*

| Traffic light icon | Description |
|---|---|
|  | A green circle icon indicates excellent status, with metric being on-target or better. |
|  | A yellow diamond icon indicates average status, with the metric being off-target but within an acceptable range. |
|  | A red square icon indicates poor status, with the metric off-target and unacceptable. |
|  | This image indicates missing data that makes it impossible to compute status. |

## Trend indicator

A trend indicator shows how the value of one column compares to the value of another column.

Trend indicators convey whether performance is improving or getting worse, not going up or going down. You can determine performance at a glance without having to decide whether an increase is good or bad.

*Table 10. Metric indicator trend icons*

| Trend icon | Description |
|---|---|
|  | A green upward facing arrow indicates that the trend value is improving in comparison to the previous period.<br>For example, a sales value is greater than the previous month or quarter. |
|  | A gray dash icon indicates that the trend value is unchanged. |
|  | A red downward facing arrow indicates that the trend value is worsening in comparison to the previous period.<br>For example, a sales value is less than the previous month or quarter. |
| Blank cell | A blank cell indicates that the trend is missing data for that period. A trend cannot be displayed when there is an incomplete status. For example, a trend cannot be displayed for the first time period, such as Q1 (quarter one). Previous data does not exist, even if the metric has a value for Actual, Target, Score, and Status. |

# Chapter 6. Model in Planning Analytics Workspace

IBM Planning Analytics Workspace includes a modeling environment that you can use to model user data with cubes, dimensions, hierarchies, attributes, and security for IBM Planning Analytics.

**Note:** Planning Analytics Workspace modeling is supported on IBM Planning Analytics Local 2.0.0 or later. Modeling is not supported in Planning Analytics Workspace on TM1 databases that are installed with version 10.3.0 or earlier.

Modeling capabilities are not available in Planning Analytics Trial.

You can use the Planning Analytics Workspace modeling tools to convert business requirements into precise cubes, dimensions, hierarchies, and calculations so that planning and analytics outcomes make sense to business users.

- Build and maintain the structure of a financial model independently, without coding.
- Transform and load data easily into a financial model, which increases transparency and confidence in results.
- Build a step-by-step financial process for multiple users based on roles and security permissions, without coding.
- Modelers can define business logic by using an integrated development environment.

To use Planning Analytics Workspace modeling, you must log in with a user name that has the Modeler role.

Planning Analytics Workspace modeling supports the following tasks:

- Creating cubes
- Using rules
- Editing dimensions
- Managing hierarchies
- Creating attributes
- Managing security

To understand why you should use the Planning Analytics Workspace tools to model your data, consider the following benefits.

## Query performance

You can create more manageable dimensions by creating more than one hierarchy in a dimension.

For example, you can create a single time dimension. A year dimension must be the same for every year to compare data between two years. This approach gives you the ability to create a new year easily, and query performance is faster because you have only one dimension in the cube.

You can create a time dimension with multiple hierarchies that represent years and months to do yearly comparisons per month.

If you create two dimensions, one for year and one for months, you gain the ability to split the years and months across two axes and you can compare data between years.

## Flexibility

Using hierarchies to add versions creates flexibility. For example, you might need to change your organizational hierarchy for planned changes. One dimension might be the organization and you can use hierarchies in the dimension to represent the organization in the future year. This hierarchy might represent the data differently from the previous year's hierarchy. With multiple hierarchies that represent the organization, you can roll up the data in multiple ways.

© Copyright IBM Corp. 2009, 2025

Hierarchies are named and contain members. You can reuse the same consolidated members in multiple hierarchies. You can use hierarchies to group these members without the need for specific consolidated names.

### Standards

Hierarchies conform to OLAP Industry standards. Planning Analytics Workspace modeling uses MDX and TM1 REST APIs to access TM1 data. The TM1 REST APIs support the hierarchy model and follow the ODATA standard.

## The modeler role

If you are enabled as an Administrator or a Modeler in Planning Analytics Workspace, you can design dimensions, hierarchies, views, and attributes to define the business logic for your application.

Your role is set when you are added to Planning Analytics Workspace by an administrator. If you are enabled as a Modeler, you can use all of the capabilities of an Analyst plus the modeling capabilities. For more information about adding users and setting their roles, see "Administer users and groups" on page 873. To learn more about roles and capabilities, see "User roles" on page 415.

To use the modeling capabilities, log in to Planning Analytics Workspace with a user name that has a role of **Administrator** or **Modeler**.

When you log in to Planning Analytics Workspace from an iPad, you are always working in Consumer mode, therefore you cannot use the modeling capabilities. For more information, see "Accessing Planning Analytics Workspace from Apple iPad" on page 427.

## Steps to building a model

You design a model in response to a business need. For example, a company wants to plan its expenses for the next 12 months by expense line. The company collects the information from a number of departments that are spread across a wide geographical area.

A model has the following basic building blocks: dimensions, cubes, and links.

Multidimensional cubes are central to the model. You create views from the cubes so that you can see just the information that you need, and you can have many different views of the same cube. For example, a regional manager might want to review data at a consolidated level, and a department manager might want to input the detailed data for their department. They use the same cube but see different views.

Dimensions give the cubes structure. A cube must have a minimum of two dimensions, but the total number of dimensions a cube can have is determined by its use. A cube that stores data can have many dimensions. A cube that is viewed by a user should have enough dimensions to define the data, but not so many dimensions that the cube is difficult to navigate. You can use hierarchies to reduce the number of dimensions in a model, while giving you the option to see alternative rollups of data.

You can quickly add business logic such as links, by creating rules.

You make the data available to other colleagues by creating books and adding views, scorecards, visualizations, graphics, and videos, and then sharing the book with them.

### Design dimensions

To make the data available for input and analysis, you must first create dimensions. Some examples of dimensions are Chart of Accounts, Products, Time, and Versions.

Find out more in "Dimensions" on page 740.

### Create hierarchies

Use hierarchies to simplify your model design. A hierarchy acts as a virtual dimension, enabling you to see alternative rollups of the same view without creating additional dimensions. This makes a model easier to maintain.

Find out more in "Hierarchies" on page 769.

### Create cubes

Use dimensions to build cubes. A cube is a store of data within a model. It is multidimensional and contains rows, columns, and any number of pages. Some examples of cubes are sales planning or expense analysis.

Find out more in "Cubes" on page 778.

### Create rules and processes

You can create rules for calculations, and processes for managing and maintaining the model. Processes can then be grouped into chores for coordinated and automated execution.

Find out more in "Rules" on page 794, "TurboIntegrator processes" on page 800, and "TurboIntegrator chores" on page 828.

## Use the modeling workbench to create and manage your model

The modeling workbench is available in Planning Analytics Workspace new experience. You cannot use the modeling workbench in Planning Analytics Workspace Classic.

To create a new workbench, click the **Home** menu ☰, then click **New** > **Workbench**.

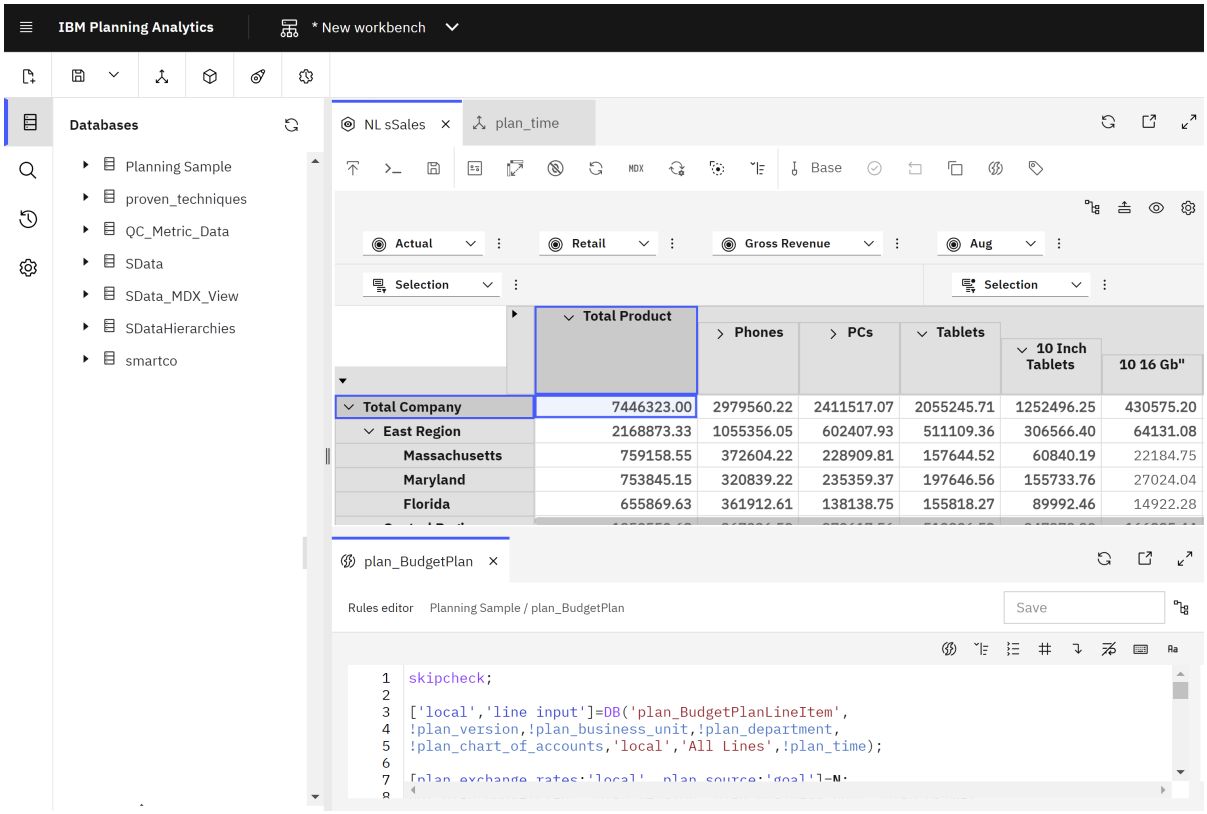
You can also create a modeling workbench on the Data and Models page. From Planning Analytics Workspace home page, Click the **Data and Models** tile, then click **New** > **Workbench** on the Data and Models page.

To open an existing workbench:

1. Click the **Data and Models** tile on the Planning Analytics Workspace home page.
2. On the **Data and Models** page, click either **Shared** or **Personal**, depending upon where the workbench is saved.
3. Click the workbench you want to open. To make it easier to find your workbench, click the **Filter** icon ▽ and select **Workbench**. You can also search for a workbench.

   Currently, modeling workbenches are visible in the **Shared** or **Personal** folders only when you open a folder from the **Data and Models** page or in the Workbench perspective.

The workbench efficiently presents your modeling assets in a tabbed and sectioned layout.




You can use the modeling workbench to:

- Create, edit, and delete any of these objects: dimensions, hierarchies, sets, processes, chores, and rules
- Run processes and chores
- Create and save cube views
- Edit settings for cubes, dimensions, processes, and chores
- Create drill processes, import data, import dimensions, reload/unload cubes
- Refresh security, set cell security

# Working in the modeling workbench

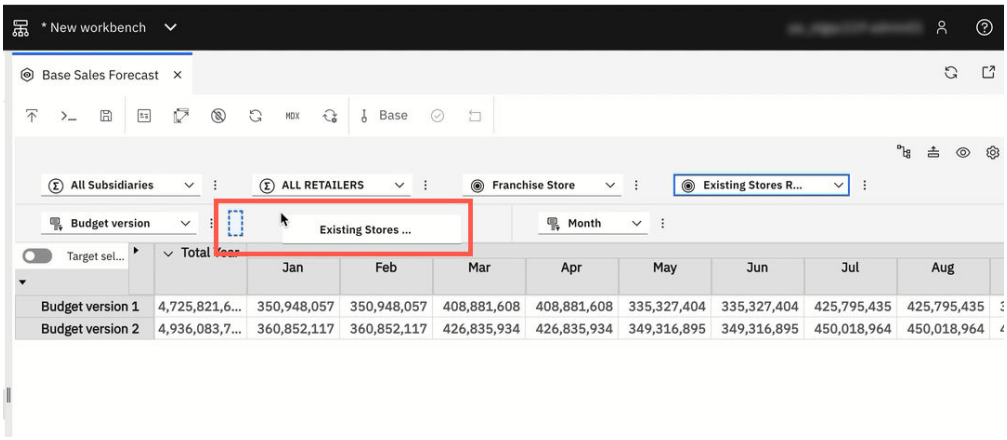
The modeling workbench includes the cube view (or exploration) editor and set editor.

## Using the view

The cube view offers flexible options to view, analyze, and organize data.

- **A guided and intuitive experience**

  For example, the view guides you to the drop zone when you drag a dimension to a different location in the grid.

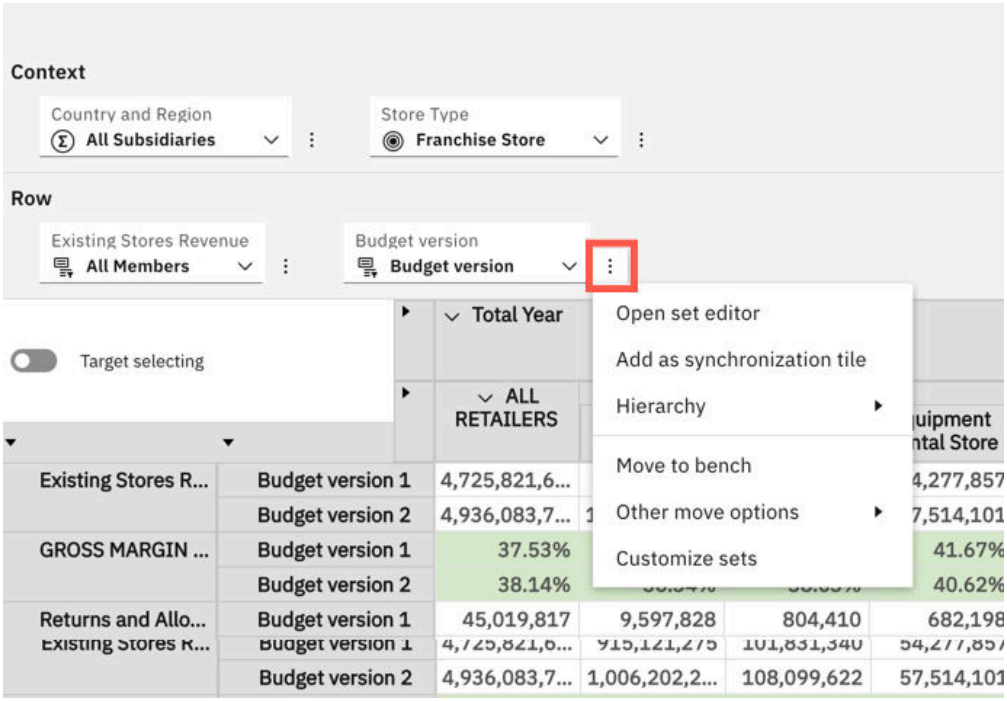



You can use the icons in the view  to:

- See the cube in the data tree.
- Expand or collapse the overview.
- See benched dimensions.
- Select hierarchy, iconography, context information, and section headers to customize your view.

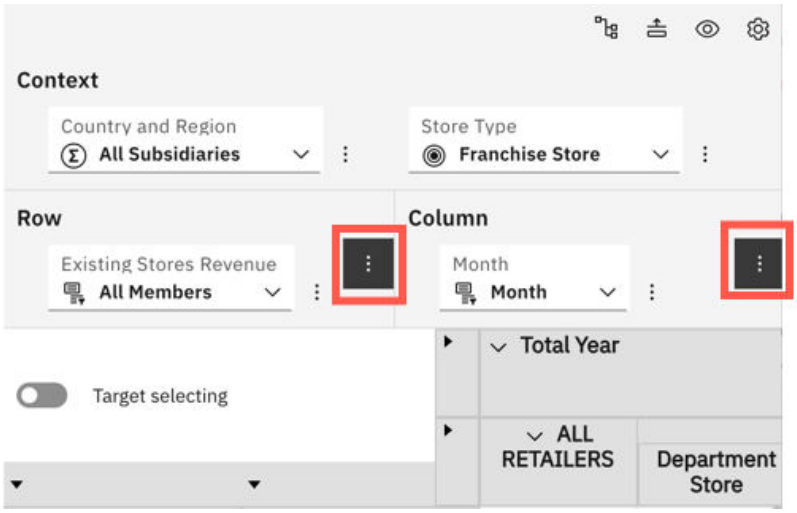
Each dimension tile in the view has a menu icon next to it from which you can:

- Open the set editor.
- Remove, replace, or add related hierarchy.
- Move the dimension to the bench.
- Manually move the dimension right or left, or to a row, column, or context.
- Limit the number of sets for a dimension (**Customize sets**).



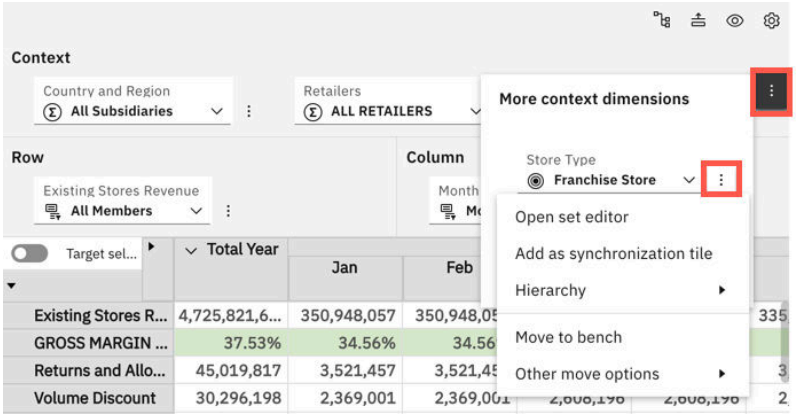

Depending on the number of dimensions or the size of the view, all dimensions might not display.

To see dimensions that are hidden from the view, click the **More dimensions** menu in the **Context**, **Row**, or **Column** area.



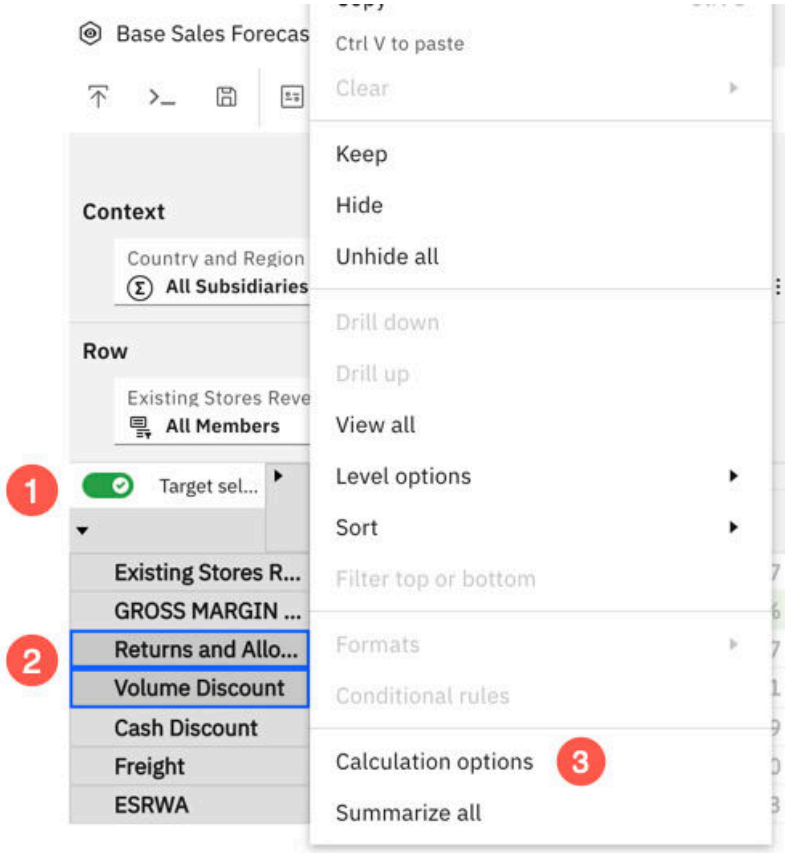

From the **More dimensions** menu, you can drag dimensions to the **Context**, **Row**, or **Column** area and vice versa. You can also move dimensions that appear within the **More dimensions** menu.

You can click the menu next to the member's name to open the set editor, move the dimension to the bench, see move options, and more.



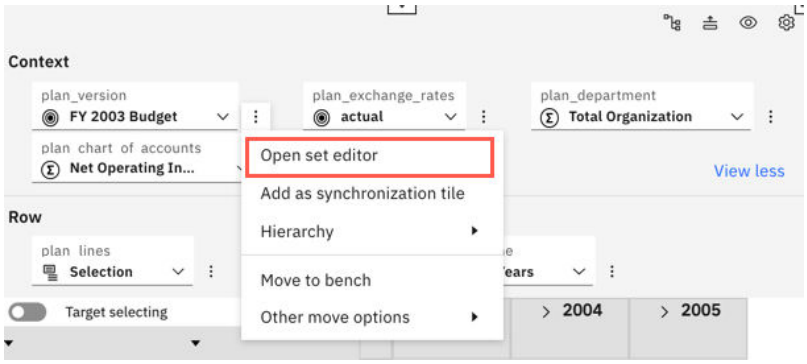

- **Target selecting toggle for asymmetric calculation**

  The **Target selecting** toggle is used to create an asymmetric calculation. Enable **Target selecting** and select the members that you want to use in the asymmetric calculation. Right-click a selected member and click **Calculation options**.

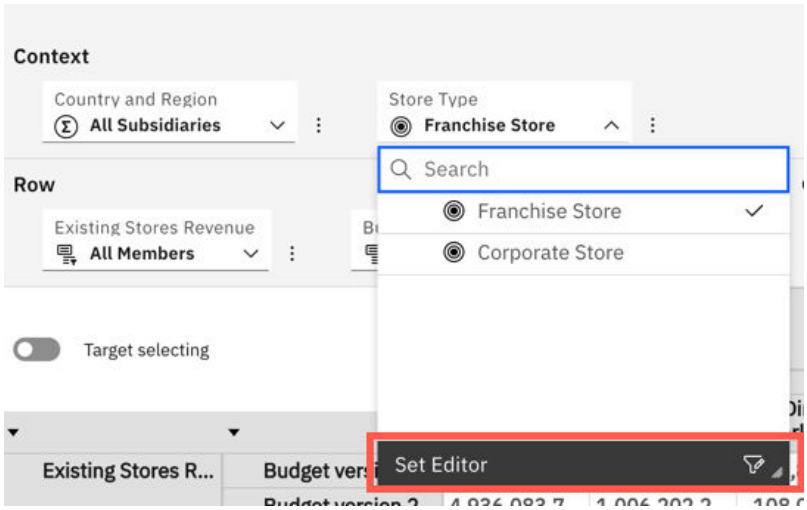



## Using the set editor

To open the set editor from the view, click the menu icon next to a dimension in the view and select **Open set editor**.




You can also double click the dimension tiles or click the **Set Editor** icon in the dimension menu to open the set editor.

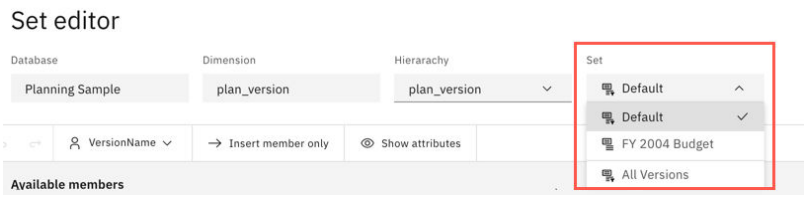



**Note:** When you open or create a set from the data tree, the set editor opens on a new tab on the active tab set of the workbench. To optimize your workspace, you can open the set editor in a new window or move the set editor to a new region on the workbench.

As with the view, the set editor offers an improved flow, a more intuitive experience, and flexible options to edit your set.

- The **Set** menu

  With the **Set** menu, you can change the set from within the set editor.




- **MDX panel**

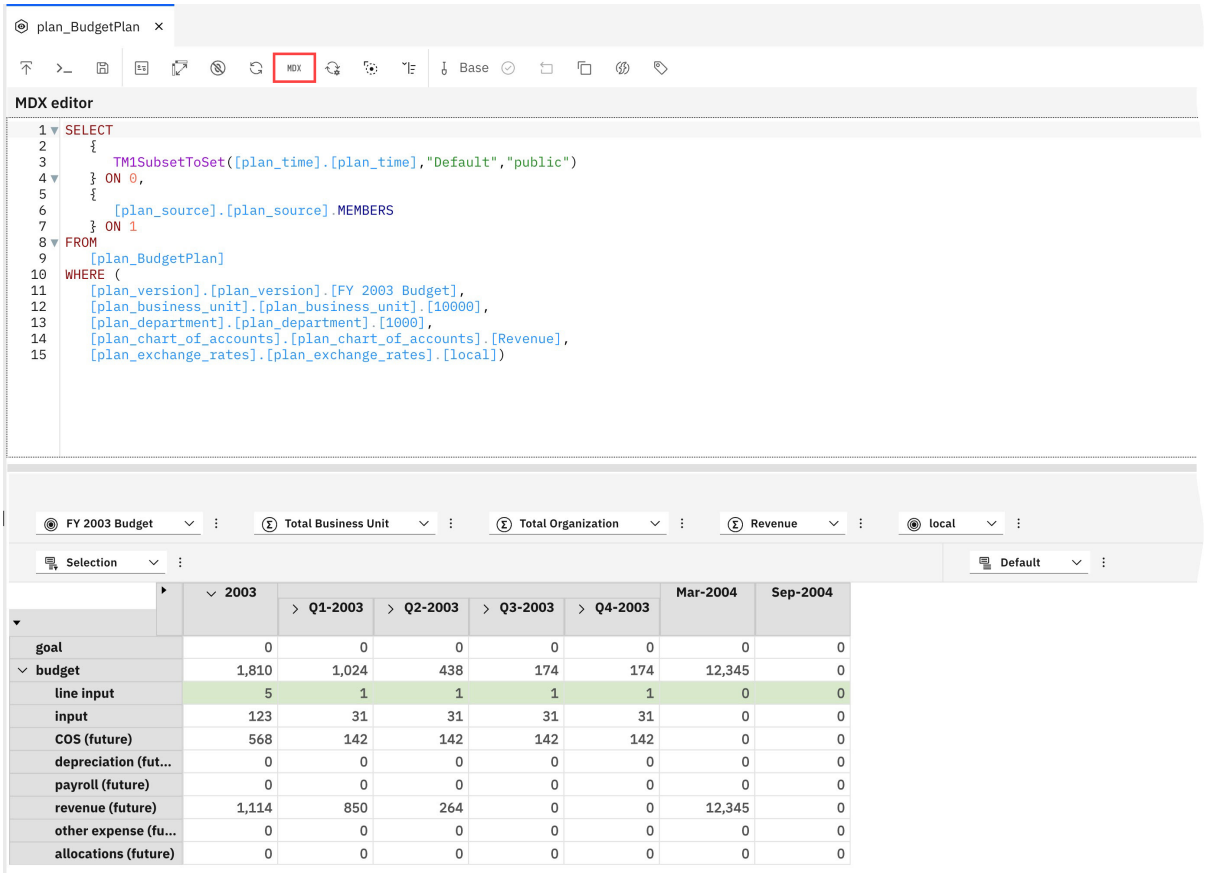
  The MDX editor allows you to make changes to the MXD expressions in a view.

  To access the MDX editor, click the **Edit MDX** icon MDX.

  The following features are available in the MDX editor:

  – Autocomplete: provides suggestions on dimension names, functions, and other TM1 objects while you are typing.
  – Syntax coloring: shows different portions of the MDX statement in different colors for increased readability. For example, keywords are written in red, and tuples are written in blue.
  – Undo: allows you to undo the last change.
  – Redo: allows you to redo the last change.
  – Find and replace: clicking ctrl+f open a dialogs that allows you to search for specific text in the MDX, and replace it.
  – Indenting: the MDX editor now automatically formats the MDX script, indenting each line. When you make changes in the view, the MDX editor updates with the correct formatting. You can manually indent the lines in the script by clicking **Indent script** in the MDX editor toolbar.
  – **Commit** applies your MDX edits to the view. When you click **Commit**, a text box appears telling you if the changes were made successfully.
  – **Synchronize** replaces the MDX in the editor with the one represented in the view. After you click **Synchronize**, click**OK** to synchronize.

**Note:** When you make edits in a view, the MDX editor automatically synchronizes to match these changes. However, when you make edits in the MDX editor, the view does not automatically synchronize to match these changes. You have to manually select where you want your edits to apply using the **Commit** and **Synchronize** buttons in the MDX editor toolbar.

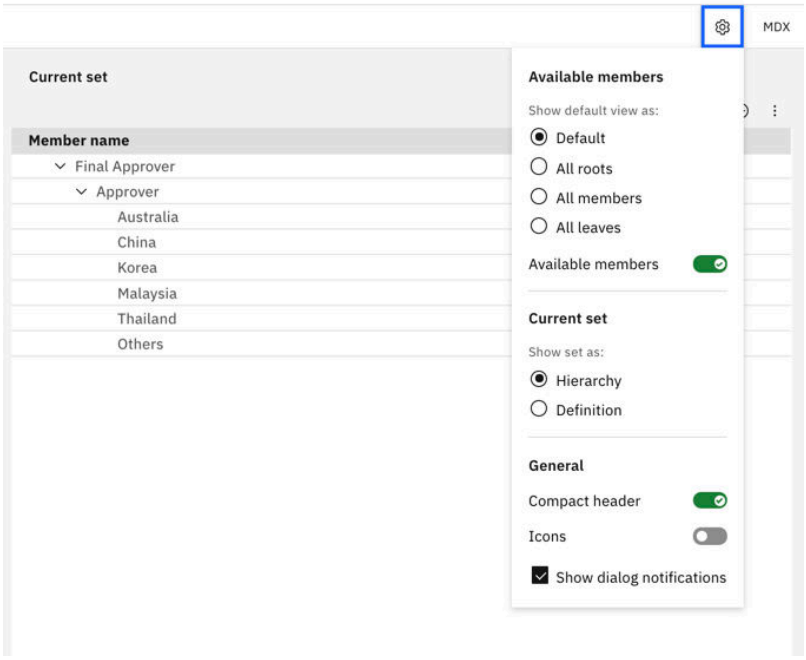



- The **Settings** menu

  The **Settings** menu offers options to customize the display for your set.

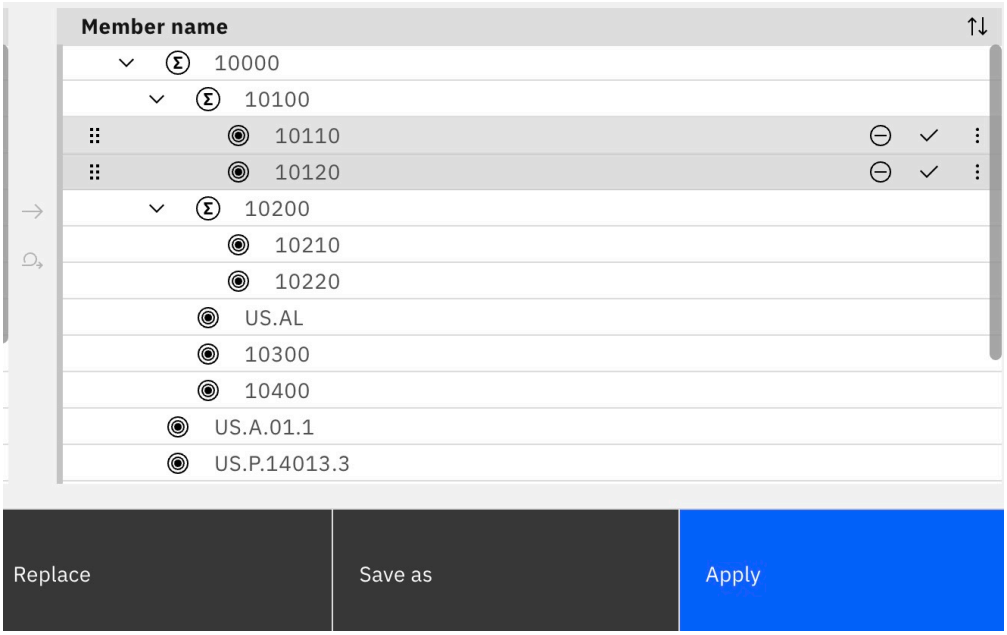
  Enable **Compact header** under the **General** section to reduce the size of the set editor header and see more members.

  You can also remove icons from all members for a simplified display with the **Icons** property under **General**.



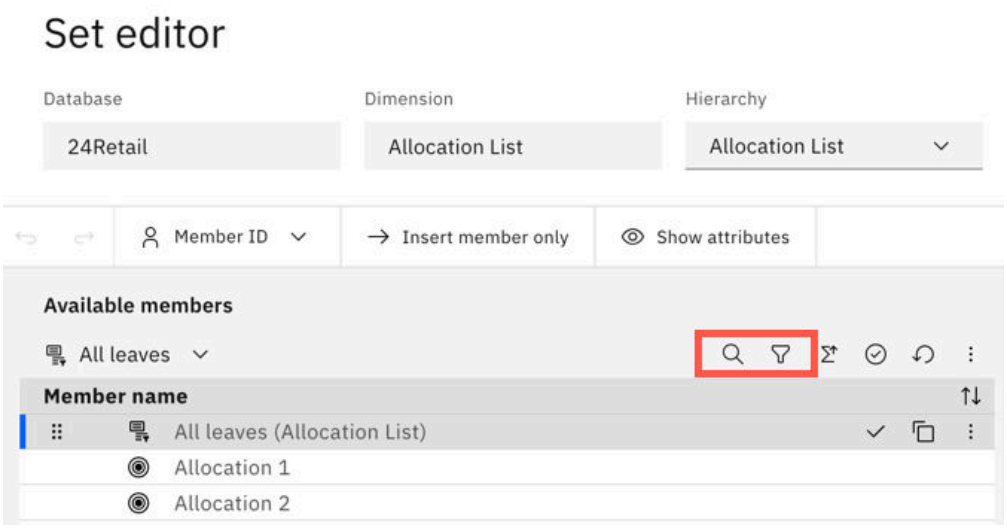

- **Multi-select activity display**

  When you select multiple members in **Available members** or **Current set**, an activity bar displays with your selection information and available options.




- **Improved features**

  The Search and Filter features in the new set editor are more intuitive and improved for usability. The filter feature allows you to filter a set by one or more attributes to choose the available members that appear in the set. The search feature lets you search for members in the set editor. You can use both features to include members in a set that match specific criteria.




The new experience set editor also offers the ability to delete sets from within the set editor.

## Create a modeling workbench

By default, a new workbench is created when you open the modeling perspective for the first time during a Planning Analytics Workspace session. There are several ways to create a modeling workbench.

### Procedure

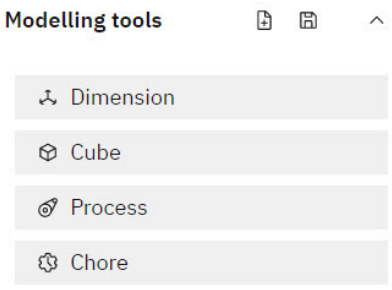
1. To create a new workbench from the Planning Analytics Workspace home page, click ☰ > **New** > **Workbench**.
2. You can also create a modeling workbench on the Data and Models page. From Planning Analytics Workspace home page, click the **Data and Models** tile, then click **New** > **Workbench** on the Data and Models page.
3. When in the modeling perspective, click **New workbench** on the **Modeling tools** bar to create a new workbench.

## Work with assets on the modeling workbench

There are multiple ways to open and work with assets on the workbench.

### Procedure

1. To create a new cube, dimension, process, or chore. click the appropriate asset on the **Modeling tools** list.




You can also create new assets from the data tree. Right-click the appropriate node, then click **Create asset_type**. For example, **Create cube**, **Create dimension**, **Create set**, and so on. This list shows the paths to create all asset types available on the data tree, with bold text at the end of the path indicating the right-click selection.

**Cubes**
Cubes > **Create Cube**

**Rules**
Cubes> [Cube name] > **Create business rules**

**Dimensions and hierarchies**
Cubes > [Cube name] > Dimensions > [Dimension name] > **Create hierarchy**

Dimensions > **Create Dimension**

Dimensions > [Dimension name] > **Create hierarchy**

**Sets**
Cubes > [Cube name] > Dimensions > [Dimension name] > **Create Set**

Cubes > [Cube name] > Dimensions > [Dimension name] > [Hierarchy name] > **Create Set**

Dimensions > [Dimension name] > **Create Set**

Dimensions > [Dimension name] > [Hierarchy name] > Sets > **Create Set**

Dimensions > [Dimension name] > [Hierarchy name] > **Create Set**

**Processes**
Processes > **Create Process**

**Chores**
Chores > **Create Chore**

2. To open and edit an existing asset, right-click the asset in the data tree, then click **Edit asset_type**. For example, **Edit dimension**, **Edit process**, **Edit business rules**, and so on.

   When you initially open or create an asset, the editor for the asset is inserted as a new tab on the active tabset within the workbench. However, you can move editors to new regions on the workbench to create an editing environment that suits your needs.

   Alternatively, you can drag and drop an exiting asset from the data tree directly onto the workbench. In this case, you can choose where to drop the asset.

   For details on using the modeling editors, see the related information links.

"Create and edit chores" on page 829
You can create chores to automate the execution of TurboIntegrator processes at defined intervals.

"Create a cube" on page 780
You can create cubes when you are a Modeler or an Administrator. Cubes are created and stored on a specific database.

"Create a dimension in Planning Analytics Workspace" on page 744
You can create and edit dimensions when you are a Modeler or an Administrator.

"Work with hierarchies in Planning Analytics Workspace" on page 769

"Create and edit processes" on page 800
You can create or edit a TurboIntegrator process in the process editor on a modeling workbench.

"Create and edit rules" on page 795
You can create and edit business rules when you are a Modeler or an Administrator.

"Create and edit a set" on page 635

A set is a collection of members in a dimension that limits the number of members that are displayed in a view.

# Organize editors on the workbench

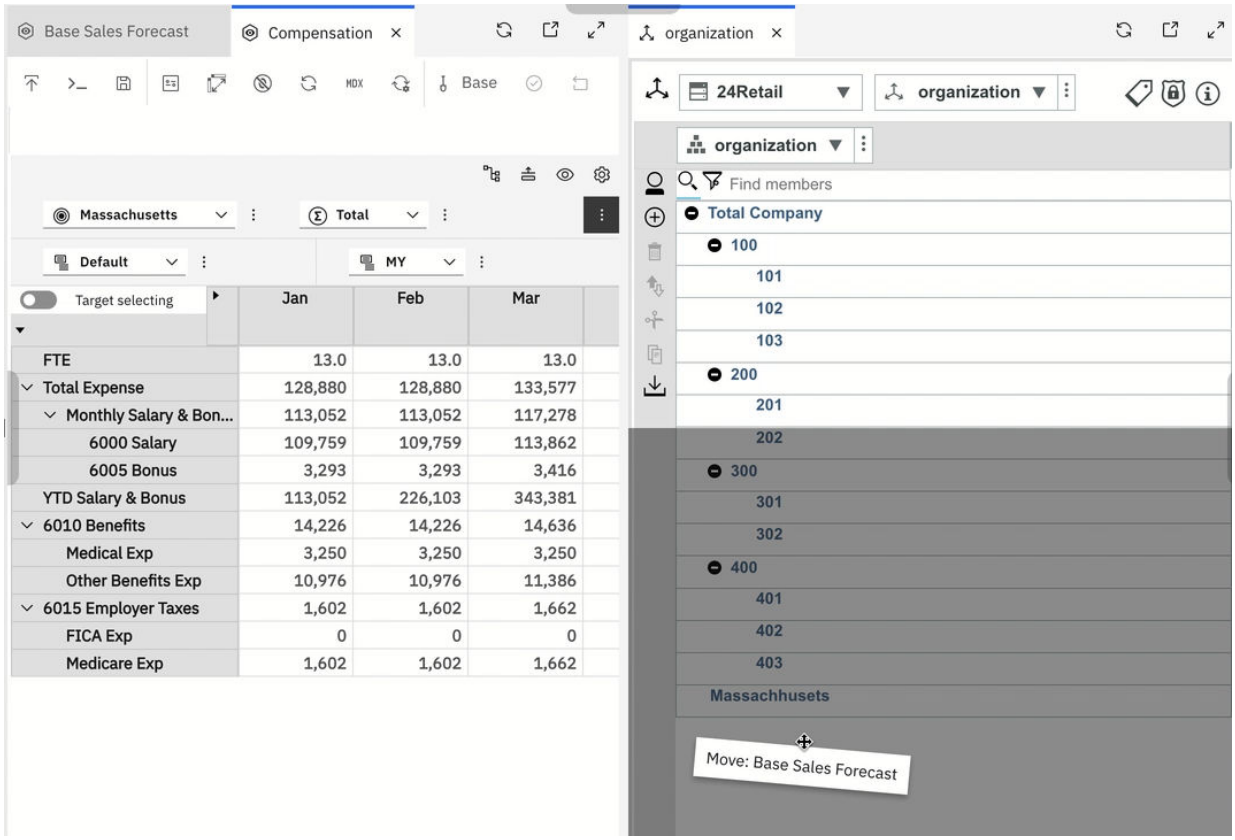
You can move editors to different tabsets on the workbench to optimize your modeling environment.

## About this task

When you initially open or create an asset, the editor for the asset is inserted as a new tab on the active tabset of the workbench. You can move the editors to new tabsets or rearrange the tabs on an individual tabset. You can also maximize an individual tabset to provide more working space.

## Procedure

1. To rearrange editors on an individual tabset, drag and drop the tab for an editor to a new location on the tabset. A vertical bar indicates the new position for the tab.
2. If a tabset on a workbench contains at least two tabs, you can drag a tab from the tabset and drop it on a different region of the workbench to create a new tabset. As you drag the tab around the workbench, grey highlighted regions indicate the locations where you can drop the tab to create a new tabset.




3. If a workbench contains multiple tabsets, it can be difficult to work in some editors. You can click **Maximize tabset** on any tabset to expand the tabset to occupy the full workbench space. After you're done working, click **Restore tabset** to return to a view of all tabsets.

   You can also open any editor in a separate window.
4. To open duplicate tabs for the Cube viewer and the File manager in the workbench, click **Duplicate**.

   The duplicate tabs point to the same TM1 object as the original tab. Use duplicate tabs to view or edit the same object in parallel without browsing away from your original place in the object. Duplicate tabs do not duplicate the underlying TM1 object, rather, they open another view of the object.

# Open editors in a separate window

You can open any editor in a separate window that is detached from the modeling workbench. This is helpful when you want to fully maximize the size of the editor. A separate window can contain only one editor.

## About this task

The functionality of all editors remains the same, whether an editor is opened in a separate window or used within the modeling workbench.

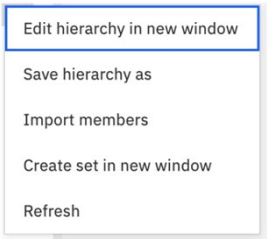
Note that when you close a modeling workbench, any editor that is open in a separate window remains open and functional. The separate window remains functional as long as you are logged in to Planning Analytics Workspace.

## Procedure

1. To open an editor in a separate window, click the tab for the editor, then click **Open tab in floating window** ⧉.

   **Important:** If an editor contains unsaved changes, these changes will be lost if they are not saved prior to opening an editor in a separate window. You will receive a warning if you attempt to detach an editor with unsaved changes, allowing you to cancel the detachment and save your changes.

   You can also open an editor in a separate window directly from the data tree. The option to open a separate window is available for dimensions, processes, rules, settings, views, chores, and sets. Press the **Ctrl** key while right-clicking on the desired asset in the data tree. You'll see options to open or create in a new window.

2. To return the separate editor window to the workbench, click **Return to workbench**.

   **Important:** If an editor in a separate window contains unsaved changes, these changes will be lost if they are not saved prior to reattaching the window. You will receive a warning if you attempt to reattach an editor with unsaved changes, allowing you to cancel the reattachment and save your changes.

   You might also receive a waring from your browser when you click **Return to workbench**, asking if you want to leave the site. You can click **Leave** to complete the process of reattaching a separate window.

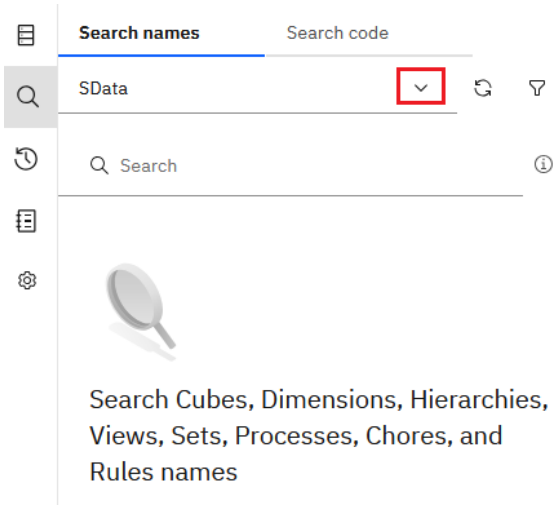
# Using database-level search in a modeling workbench

Use the search feature in the modeling workbench to search for data items in a database, and search the database processes and rules code.

## Search names

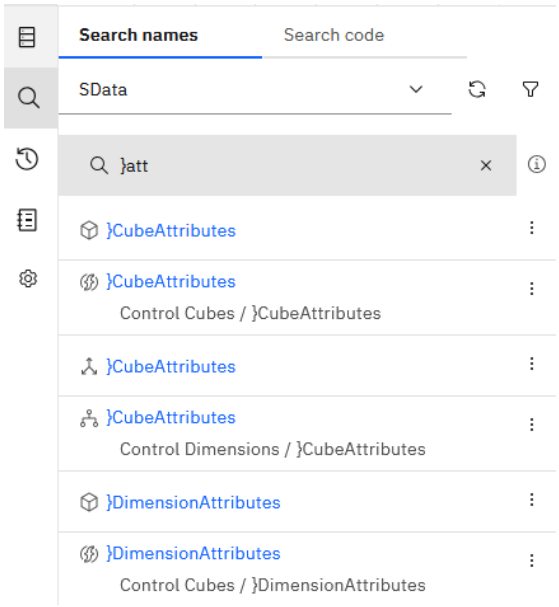
Use the search feature in the modeling workbench to search for different data items (cubes, dimensions, views, sets, processes, chores, rules, and control objects) in a database.

1. Click the **Search** button in the workbench.
2. In the panel that opens, select the database in which you want to search.




3. Enter the name of the data item you are searching for in the `Search` field.

   To search for a control object, prefix the name with `}`. For example, `}Attributes`. The search looks for items that contain the string that follows `}`. A search for `}att` displays the results }CubeAttributes and }DimensionAttributes.




   **Tip:** Click the **Filter** icon to narrow down the search results to specific data items. For example, if might want to see search results only for dimensions.

   To search for the same data item in a different database or with different filter options, simply change your selections, which triggers a new search.

4. Click any search result to open the item in the content panel.

   You can also click the context menu icon ⋮ next to the data item to edit it or delete it from the database.

   You can make the following changes to each data item from within the search panel:

   - **Cubes**
     - Add new view
     - Import data
     - Delete cube

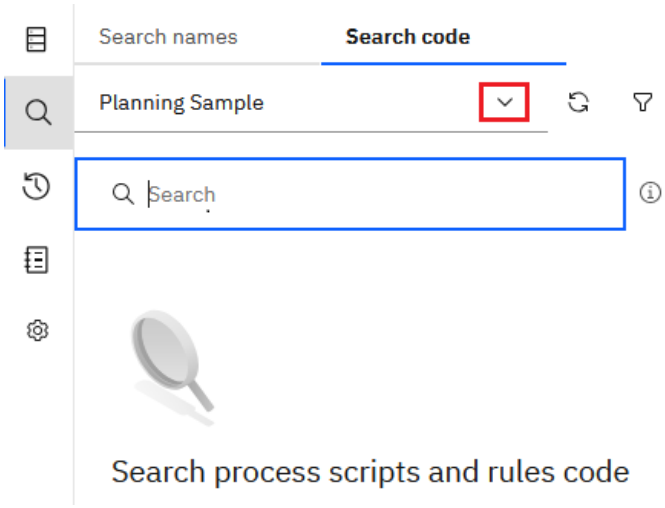
  - Edit cell security (not available for control cubes)
- **Rules**
  - Edit business rules
  - Delete business rules
- **Dimensions**
  - Edit dimension
  - Create set
  - Delete dimensions
  - Create hierarchy (not available for control dimensions)
- **Sets**
  - Edit set
  - Delete set
- **Views**
  - Add view
  - Delete
- **Processes**
  - Edit process
  - Run process
  - Delete process
  - Display chores involved
- **Chores**
  - Edit chore
  - Run chore
  - Delete chore
  - Enable chore schedule

## Search code

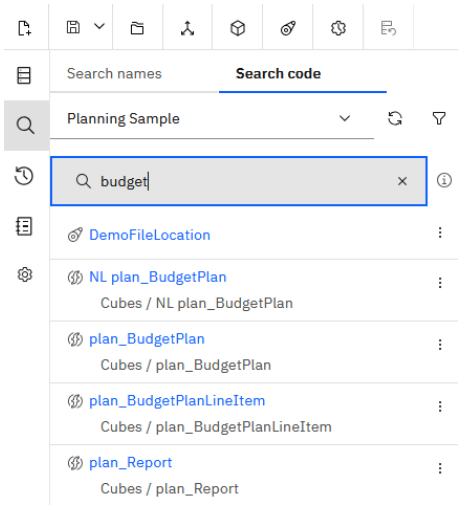
Use the **Search code** tab to search the process or rule code in the TM1 database.

**Note:** The control processes code is not searched in this release.

1. In a workbench, click the **Search** button in the vertical toolbar, and then click the **Search code** tab.
2. In the panel that opens, select the database in which you want to search.




3. In the `Search` field, enter the keywords that you are searching for, and press **Enter**.

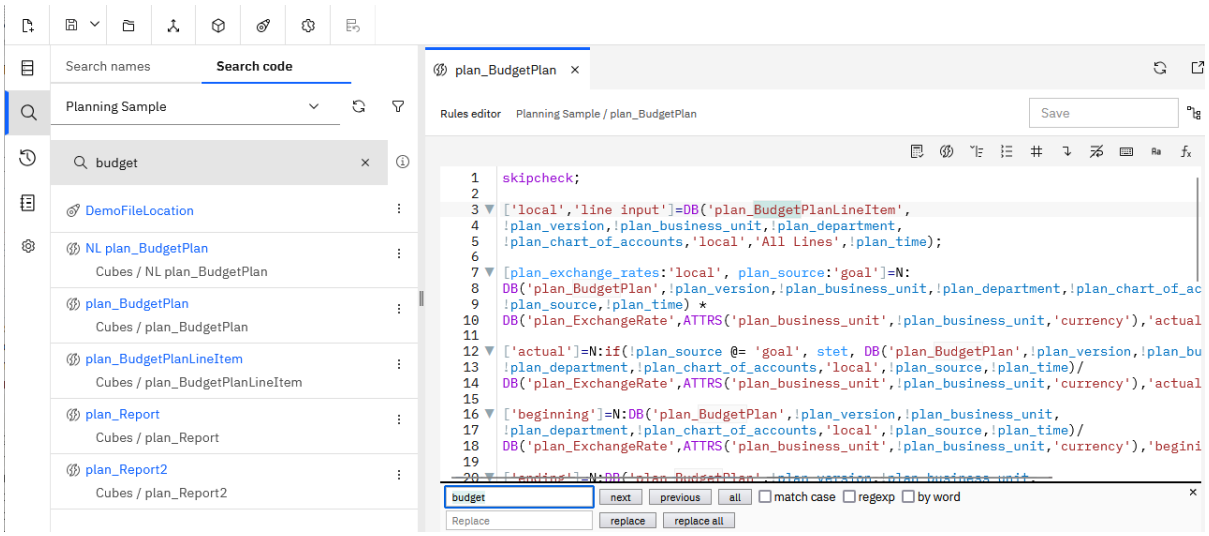



**Tip:** Click the **Filter** icon to narrow down the search results to either processes or rules.

The search results include the processes and rules that contain the specified keywords.

4. Click any of the results.

The rule or process editor is opened with the first instance of the keyword highlighted in the code. Use the buttons in the bottom panel of the editor to locate other instances of the keyword in the code.



# Workbench preferences

Enhancements to user **Preferences** are available across multiple components.

## Design update

A few design changes have been made to **Preferences**.

The **Preferences** icon is now ⚙. This button is always visible, even when the workbench is empty..

The order of sections and their sub-sections in **Preferences** is as follows:

- **Database tree**
  - **Display members**
  - **Hide Databases**
- **Dimension editor**
  - **Show dimension editing warning**
  - **Compact header**
  - **Value bar**
  - **Attribute type**
  - **Element index**
  - **Contex menu**
  - **Page size**
- **Cube viewer**
  - **Show hierarchy name**
  - **Show context information**
  - **Show icons**
  - **Show section header**
- **Process editor**
  - **Font**
  - **Font size**
  - **Enable line wrapping**
  - **Lock generated statements**
- **Rules editor**
  - **Font**
  - **Font size**
  - **Enable line wrapping**

By default, all sections are collapsed, but retain their previous state when the user opens or closes the **Preferences** panel. Opening and closing **Preferences** also refreshes the selected values.

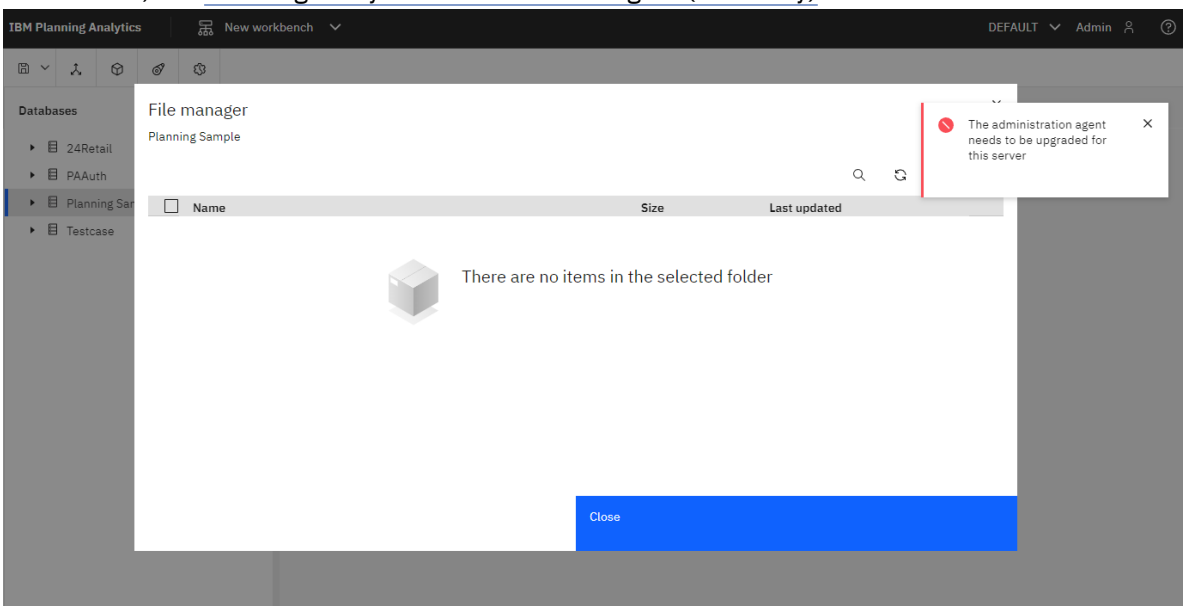
**Note:** The selected preferences apply only to the workbench, not the dashboard.

# Using the File manager

The **File manager**, a tool on the modeling workbench, shows a list of all the files and folders that have been uploaded to the Planning Analytics database and allows you to manage those items. You must be an Administrator or Modeler to work with **File manager**.

**File manager** is available only on the modeling workbench; you cannot open **File manager** from a book.

**Attention:** If you're using IBM Planning Analytics Workspace Local, update the Planning Analytics Administration agent to the version that is included in Planning Analytics Workspace 2.0.95. You won't be able to use the **File manager** in Planning Analytics Workspace 2.0.95 unless you update the Planning Analytics Administration agent to the version from the same release. For more information, see Planning Analytics Administration agent (local only).

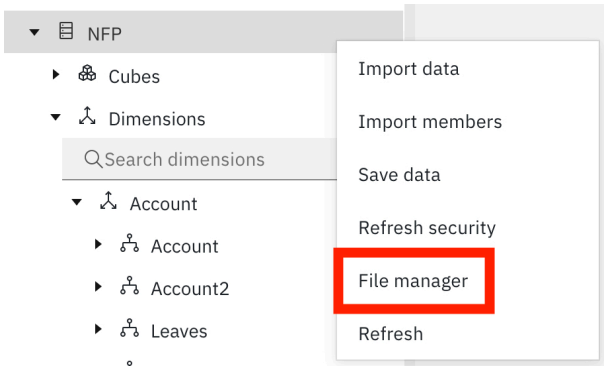



With **File manager**, you can complete the following tasks:

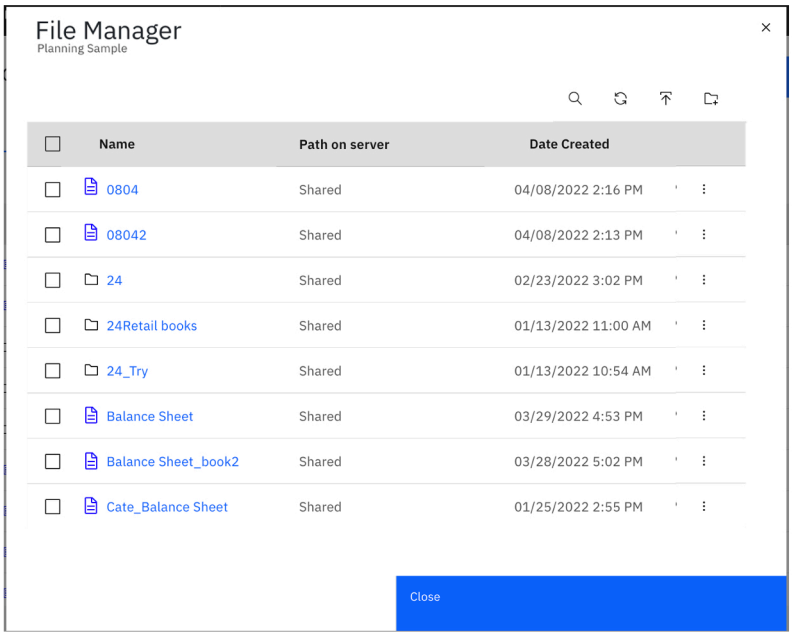
- Upload a file, or multiple files at once
- Delete a file, or multiple files at once
- Create folders and sub-folders *
- Organize uploaded files into folders using Cut, Copy, and Paste *
- Compress and expand files *
- Delete folders
- Download files
- Refresh files

* Not available in Planning Analytics Workspace on Planning Analytics as a Service or when connected to TM1 Database 12.

To launch **File manager** for a database, right-click the database in the Databases tree on a modeling workbench, then click **File manager**.




**File manager** displays a list of all the files and folders that have been uploaded to the server.




## Working with files and folders in File manager

You can use the toolbar to create folders and sub-folders, upload files, refresh files, and search for a folder or file. Use the **Actions** button ⋮ or select a file or folder to access options such as Download, Delete, Cut, Copy, and Compress.

### Deleting a file or folder

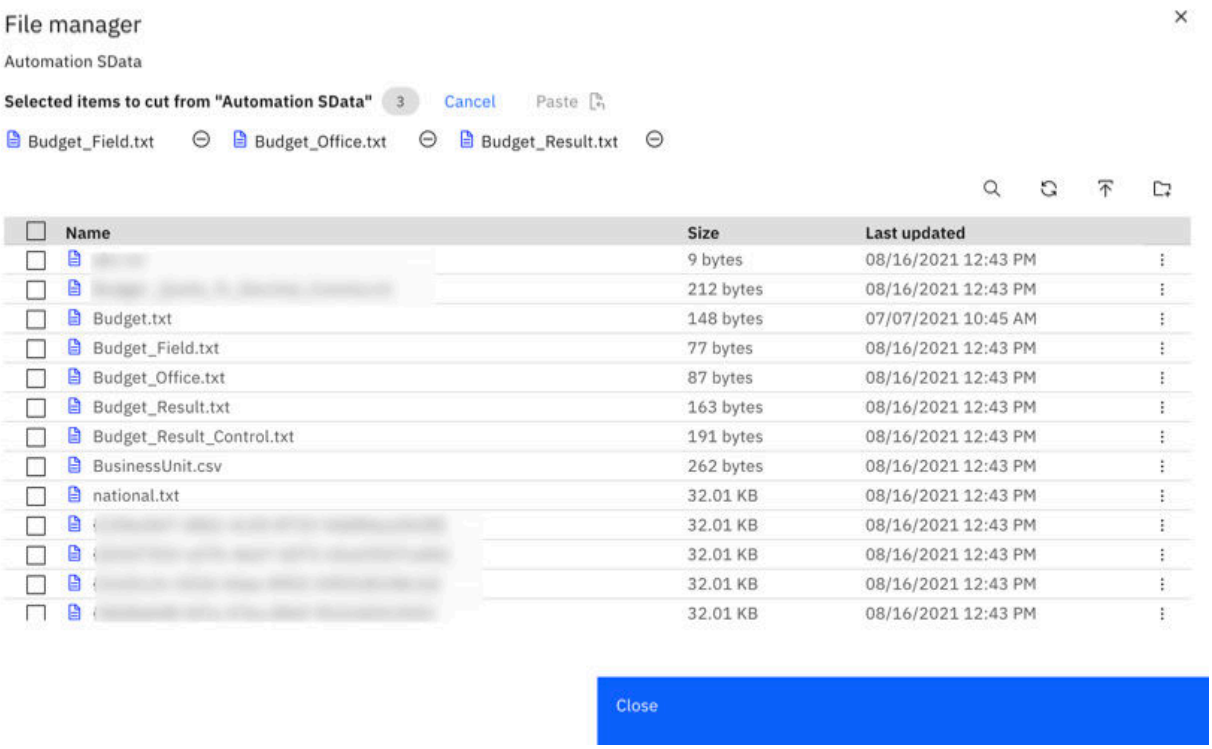
To delete an item, select the file or folder and click **Delete**. You can also click the Action button for the item and click **Delete**.

**Warning:** When you delete a folder or sub-folder, all files within that folder are also deleted.

### Cut, copy, and paste files

You can move or copy files or folders to a different folder within the **File manager**. The **Cut** and **Copy** options are available from the **Actions** button or display in the toolbar when you select one or more items.

To cut or copy, select one or more items and click **Cut** or **Copy**. Your selections display in alphabetical order as a horizontal list for a quick review before you cut or copy the items.




After you review the list of items that you want to cut or copy, you can paste the items to a different folder. To paste a cut or copied item, go to the destination folder and click **Paste**.

**Compress and expand files**

Select one or more files or folders and click **Compress** to compress the items into a .zip file. You can compress a file and move it to a new location without using **Cut**. To do this, select a file and click **Compress**. Then, go to the destination folder and click **Compress** again.

To expand a compressed file, select the file and click **Expand**. As with compressing a file to a new location, you can expand a file to a new folder.

## Using the Database explorer

The **Database explorer** is a tool on the modeling workbench that shows information on database objects and allows you to perform common bulk actions on those objects. You must be an Administrator or Modeler to work with **Database explorer**.

To open the **Database explorer**, right-click the database on the modeling workbench Databases tree, then click **Database explorer**.

The Database explorer opens a Properties tab with the name of the database, which displays a table with the main TM1 objects that you can view, along with the numbers of each object in the specified database. Click the item count to open a table that displays all of the specified objects in the database. You can select multiple objects in the table and perform a bulk action on all selected objects.

With **Database explorer**, you can complete the following bulk actions on the following database objects:

- Cubes
  - Delete
  - Lock
  - Unlock
- Dimensions
  - Delete
  - Lock

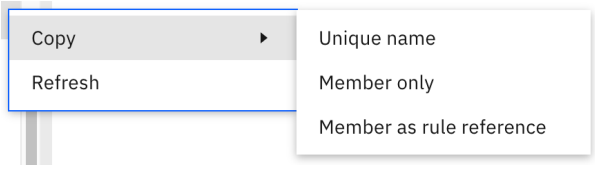
  - Unlock
  - Reserve
  - Release
- Hierarchies
  - Delete
- Sets
  - Delete
- Processes
  - Delete
  - Update ODBC data sources
- Chores
  - Delete
  - Enable chore schedule
  - Disable chore schedule
- Control objects
  - Delete

You can also use filters to look for specific database objects in the system.

## Copy menu on the modelling workbench

The **Copy** menu allows you to copy different parts of member and hierarchy scripts to the clipboard, which allows you to easily edit the MDX, rules, and TurboIntegrator processes.

The **Copy** menu is located on the context menus for TM1 objects in the tree.




To access the **Copy** menu, right-click on a member or hierarchy in the tree, click **Copy**, then select one of the following options:

- **Unique name**: Copies the unique name. This is used in the MDX.
- **Member only**: Copies the member. This is used in rules and TurboIntegrator processes.
- **Member as a rule reference**: Copies the member as a rule reference. This is used in rules.

When an option is selected, the corresponding text is copied to the clipboard.

## Change the security of a websheet

In the modeling workbench, you can modify the security of websheets and the folder structure used to store them.

**Note:** The following feature requires TM1 11.8.29 or later, or TM1 v12.

To manage the security of a websheet, complete the following:

1. Right-click on a websheet in the database tree.
2. Click **Websheet security**. The `PlanSampLD:}ApplicationSecurity` tab opens. Each public file that is part of the websheet is displayed in table format.

   **Note:** This step is for TM1 v11 only.

3. Right-click on the cell that corresponds to the file you want to change the security of. Choose one of the following options:
   a. None: Members of the user group cannot see or edit the file.
   b. Read: Members of the user group can see the file, but cannot edit it.
   c. Write: Members of the user group can read and edit the file.

## Make a websheet public or private

You can set a websheet to be public or private on the modelling workbench.

**Note:** The following feature requires TM1 11.8.29 or later, or TM1 v12.

A private websheet can only be viewed or edited by you.

To make a public websheet private, complete the following:

1. Right-click on a public websheet in the database tree.
2. Click **Security**.
3. Click **Make private**. The folder icon changes to indicate that it is now a private folder 

To make a private websheet public, complete the following:

1. Right-click on a private websheet in the database tree.
2. Click **Security**.
3. Click **Make public**. The folder icon changes to indicate that it is now a public folder 

## Create shortcuts to TM1 objects in the modelling workbench

The shortcut feature in the Planning Analytics Workspace modelling workbench supports all object types in the TM1 database, allowing you to easily group, organize, and access modelling objects. Shortcuts to TM1 objects save in the **Applications** folder in the data tree.

**Note:** This feature requires TM1 versions 11.8.29 or later, or TM1 v12.

You can create shortcuts for the following objects:

- Dimensions
- Cubes
- MDX views
- Native views
- Processes
- Chores
- Websheets
- Subsets

  **Note:** In TM1 v11, you cannot create shortcuts for subsets with hierarchy names that are different from the dimensions.

To create a shortcut to an object from the database tree, right-click the object and select **Add shortcut**. Select a destination folder or create a new folder within the **Applications** folder. If the object is already open in the workbench, you can create a shortcut from the workbench tab for each supported object type with the **Add shortcut** icon.

After you create a shortcut, you can use the object from the shortcut by double-clicking the shortcut or by dragging the shortcut to the workbench.

You can also perform the following actions by right-clicking the shortcut and selecting one of the options:

- Rename shortcuts
- Delete shortcuts

  **Note:** Deleting a shortcut doesn't delete the TM1 object. Similarly, if you delete an object from the database tree, the related shortcut isn't deleted. When you use a shortcut for a deleted TM1 object, you'll get an error notifying you that the object could not be found.
- Apply security to the shortcuts

  You can set the shortcuts to be public or private
- Open all shortcuts in the folder (folders only)

  You can open all shortcuts within a folder. Each shortcut opens as a separate tab in the workbench. This selection opens only shortcuts in the immediate folder, while shortcuts in sub-folders are not included. There are some objects that you cannot open in the workbench, such as websheets.
- Run the process (TurboIntegrator processes only)

  If the shortcut is to a TurboIntegrator process, you can run that process through the shortcut.

### Accessing TM1 object shortcuts in books

You can access all shortcuts, except those that are for processes and chores, in Planning Analytics Workspace books, from the **Applications** folder in the database tree.

## Dimensions

Dimensions are lists of related members. Two or more dimensions are used to make a cube that can be used for planning and analysis.

Typical dimensions a cube might contain are time, versions, regions, products, departments, measures. A member is an item in a dimension, so in a time dimension, you can have months, years, quarters. Each month, year, and quarter is a member.

Dimensions can be a simple list with all members at the same level, or a dimension can be structured with members at different levels and with multiple hierarchies. How a dimension is structured depends on how you want the data to be represented. You might want to have a simple time dimension that just contains a list of the months, or you might want a time dimension that is grouped by years, quarters, and months, as shown in the following list.

- 2017
  - Q1-2017
    - Jan-2017
    - Feb-2017
    - Mar-2017
  - Q2-2017
    - Apr-2017
    - May-2017
    - Jun-2017
  - Q3-2017
    - Jul-2017
    - Aug-2017
    - Sep-2017
  - Q4-2017
    - Oct-2017
    - Nov-2017

- Dec-2017

Levels define the way data is grouped in dimensions. A dimension can have a number of levels relative to their hierarchical structure, and these levels are automatically named Level000, Level001, Level002, Level003 and so on. For a dimension with a single level, the level is named Level000.

If you have a dimension that is structured with multiple levels, you can choose to show the members at a particular level. For example, you could show the leaf level, which is just the months, or you could show just the quarters, or the years. To select a level, in a cube view, click the dimension tile and then select the level.

Typically, cubes contain a measures dimension. A measures dimension contains the measures that you want to track in your business analysis. Examples of measures include sales amounts, units sold, expenses, acquisition values, and campaign costs.

You can define hierarchies for dimensions. Every dimension in IBM Planning Analytics Workspace has at least one hierarchy. You can define alternative hierarchies so that you can roll up a hierarchy in different ways without having to add extra dimensions. To find out more, see "Hierarchies" on page 769.

You can define attributes for dimension members. Attributes help to explain or describe a dimension member, and could be something like color, size, or type. You can create hierarchies from attributes by right-clicking an attribute in a dimension and clicking **Create Hierarchy**.

# Using the dimension editor

Use the dimension editor in Planning Analytics Workspace to create and edit dimensions and hierarchies.

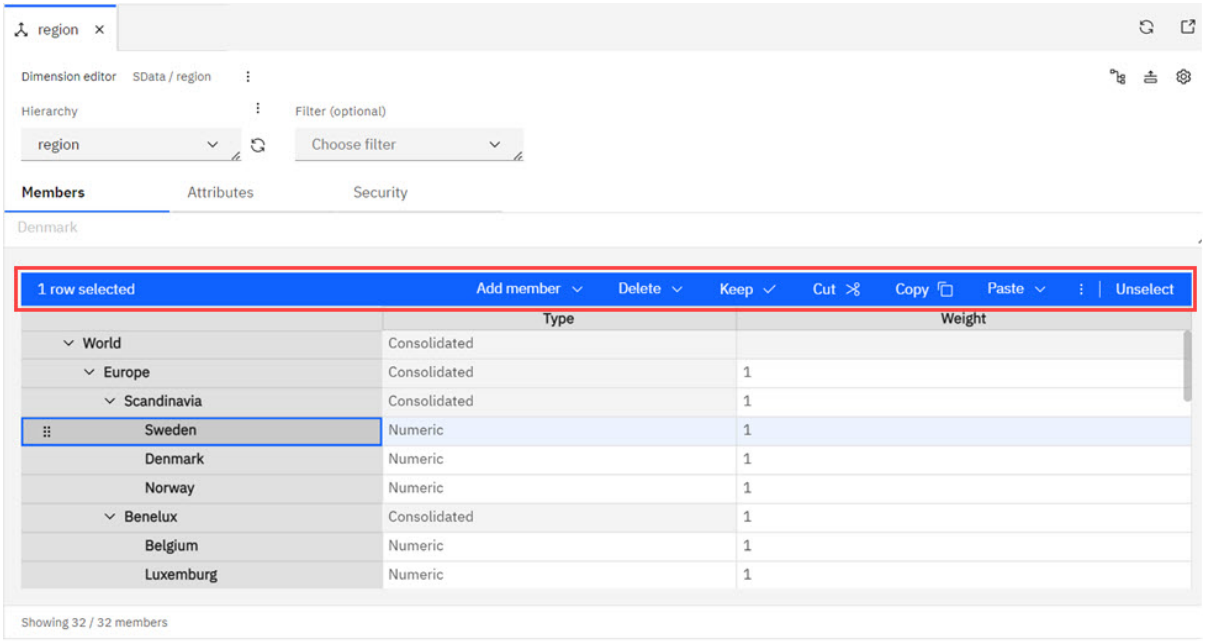
To use the dimension editor to create a dimension, right-click the **Dimensions** node in the **Databases** tree, and then click **Create dimension**.

To use the dimension editor to edit an existing dimension, right-click the dimension name in the **Databases** tree, and then click **Edit dimension**.

To create a new hierarchy for a dimension, right-click the dimension name in the **Databases** tree, and then click **Create hierarchy**.

## Working with members

When you select one or more members in the dimension editor, an action toolbar becomes available.



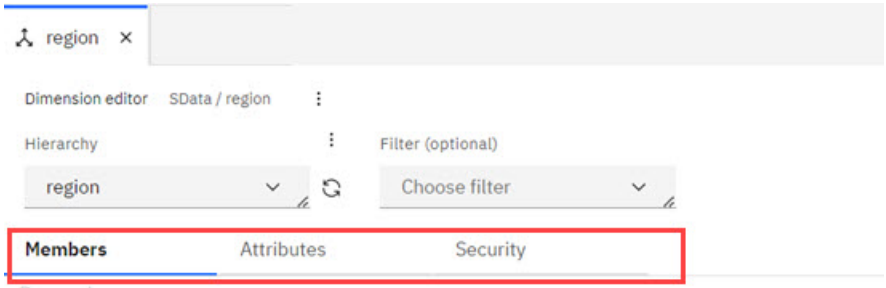
The action toolbar provides all the capabilities of the right-click (context) menu and the vertical toolbar from the classic dimension editor. Use this toolbar to:

- Add or delete members
- Change the order of members
- Cut copy, and paste members
- Hide or reveal members
- Set display formats

**Tip:** If you're adding, deleting, or otherwise modifying more than 100 members, consider using the applicable functions in a TurboIntegrator process rather than the dimension editor. A process runs more efficiently, consuming less time and resources when modifying a large number of members.

## Attributes and security

You can create and modify member attributes and security in the dimension editor. Click the appropriate tab to work with either **Attributes** or **Security**.




For more information on working with attributes, see “Member attributes” on page 771.

For more information on working with member security, see “Secure dimension members” on page 851.

## Expand or collapse the header

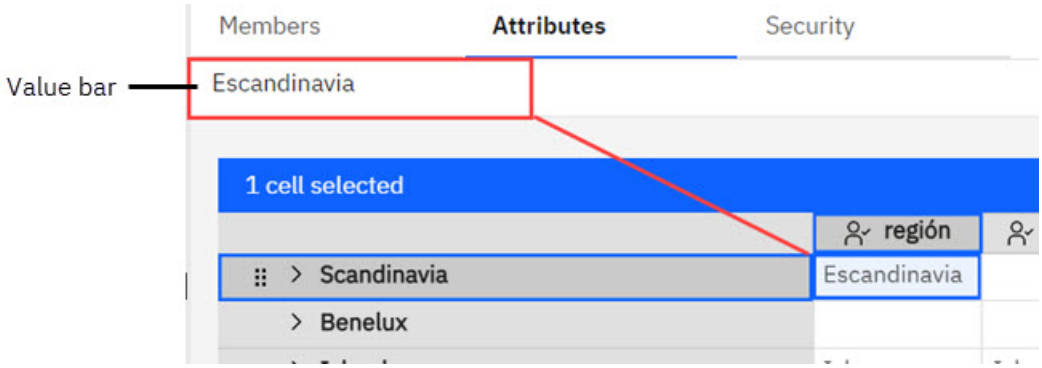
Click **Expand header** or **Collapse header** in the dimension editor toolbar to control the display of the header.

The header displays the **Hierarchy** selector and **Filter** option. Expand the header to display both the Hierarchy selector and Filter option. Collapse the header to maximize the number of members that are displayed in the dimension editor without scrolling.

## Change the view options

Click the **Settings** icon to enable or disable the following user interface elements:

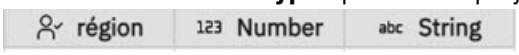
**Value bar**
: Use the **Value bar** to edit member names or cell values outside of the dimension editor grid. This editing option is helpful, for instance, when a cell is narrow but an alias name is long.




Enable the **Value bar** option to display the value bar. Disable this option to maximize the number of members that are displayed in the dimension editor without scrolling.

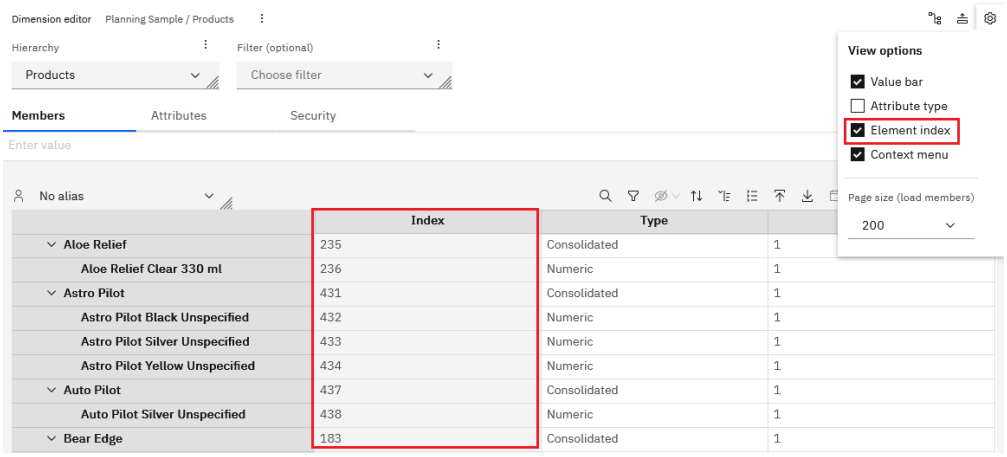
**Attribute type**

Enable the **Attribute type** option to display the data type icon for each attribute on the **Attributes** tab. Disable the **Attribute type** option to display only attribute names on the **Attributes** tab.




**Element index**

Display or hide the **Index** column in the dimension editor. The **Index** value defines the order of elements within a dimension or hierarchy, and is unique for each element.




**Context menu**

Enable or disable the right-click menu for members.

**Page size (load members)**

Select the number of members to initially load when the dimension editor is opened and each time a new page is fetched. You can load and display up to 600 members per page.

**Tip:** The same options are available from **Profile and settings**. Use this location to enable the different options by default for the entire workbench environment. For more information, see "Manage settings for components and features" on page 416.

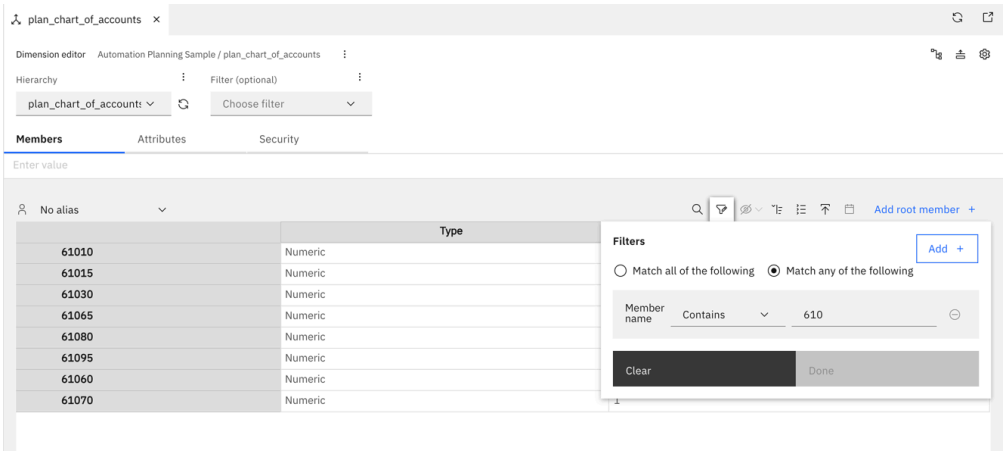
## Filter members

Use the **Filter (optional)** option to narrow the focus of members displayed in the editor. You can select any saved set or member level.

Use the **Filter (optional)** option to narrow the focus of members displayed in the editor. You can select any saved set or member level.



Click the **Filter** icon in the dimension editor toolbar to filter on member names. You can choose operators such as = or < > to create dynamic selections and combine up to three filter parameters in one go.




When you apply these filters on the **Members** tab, the same filters automatically apply to the members on the **Attributes** tab.

Clearing filters on the **Members** or **Attributes** tab, clears the filters on both tabs.

### Sort members

You can choose from several sort orders to sort hierarchy members in the dimension editor. Click the **Sort members** icon ↑↓ in the toolbar to access the following sort options:

**Ascending**
Sort members in ascending order.

**Descending**
Sort members in descending order.

**Hierarchize**
Organize the members of a specified set into hierarchical order.

**Hierarchically ascending**
Sort members within a hierarchy in ascending order.

**Hierarchically descending**
Sort members within a hierarchy in descending order.

**By index, ascending**
Sort members by index, in ascending order.

**By index, descending**
Sort members by index, in descending order.

**By index, hierarchically ascending**
Sort members within a hierarchy by index, in ascending order.

**By index, hierarchically descending**
Sort members within a hierarchy by index, in descending order.

## Create a dimension in Planning Analytics Workspace

You can create and edit dimensions when you are a Modeler or an Administrator.

### Procedure

1. On a modeling workbench, navigate to the **Databases** tree and expand the database where you want to create the dimension.
2. Right-click **Dimensions** node, and click **Create dimension**.

3. Enter the name of your dimension and click **Create**.

   The name can't contain these characters: \ / : * ? " < > | ' ; , }

   By default, a dimension is created with a an initial hierarchy that has the same name as the dimension.
4. Add members to the hierarchy by using one of these methods:
   - Click **Add root member** to start adding members manually.
   - Import members from a file ("Import members and attributes into a dimension in Planning Analytics Workspace" on page 745).
   - Add members to the dimension ("Add members to a dimension or hierarchy in Planning Analytics Workspace" on page 760).

   You can also customize the dimension as a Time dimension.

### What to do next

To create a new hierarchy within the current dimension, click ⋮ next to Hierarchy and click **Create hierarchy**. For more information about creating hierarchies, see "Hierarchies" on page 769.

To delete a dimension, in the data tree, right-click the dimension, and click **Delete dimension**.

## Import members and attributes into a dimension in Planning Analytics Workspace

### About this task

You can import members into a dimension from multiple locations on the **Databases** tree. Depending on the launch point, your ability to create or modify metadata might be limited.

If you click **Dimensions**, **Import members** on the Databases tree or use the **Import members** option on the dimension editor, there are no limitations on metadata updates. You can:

- update an existing dimension and an existing hierarchy
- update an existing dimension and create a new hierarchy
- create a new dimension and a new hierarchy

If you click **<dimension name>**, **Import members** on the Databases tree, updates apply to the dimension you clicked, but you can pick the hierarchy to update or choose to create a new hierarchy.

If you click **<hierarchy name>**, **Import members** on the Databases tree, updates apply only to the hierarchy you clicked.

### Procedure

1. Use one of the options described above to start the import.
2. As applicable on the **Import members** pages, specify the **Dimension** and **Hierarchy** to update.

   If you want to create a new hierarchy, select the **New hierarchy** option and enter a name for the new hierarchy.
3. Specify a **Data source**.
   - For a delimited text file source, select **File** from the menu.

     To import a file that is external to your Planning Analytics database, enable the **Upload file** option, then drag the file onto the drop zone or click the link to navigate to the file.

     **Tip:** In IBM Planning Analytics Workspace Local, drag and drop requires the Planning Analytics Administration agent to be configured and running.

To import data from a file that is stored in File Manager on your Planning Analytics database, disable the **Upload file** option, then click **Select file**.

- For an ODBC source, select **Database connection**. Select the connection you want to use from the **Database connection** list, then provide authentication credentials if required. If the database supports Unicode, enable the **Unicode** option.

  TM1 Database 12 requires the ODBC Data Connector to use a database connection for a data source.

4. Click **Next**.
5. Preview the data source.
   - For a file source, preview the file and confirm that the correct **Delimiter**, **Quote character**, and **Header records** formatting options are applied. If necessary, modify any of the formatting options and then click **Refresh** to see the impact on the preview.
   - For an ODBC source, enter a query to extract data, then click **Refresh** to preview the results.
6. Click **Next**.
7. Select the **Mapping type** that you want to apply to the import.

   **Leaf only**
   Imports a flat list of leaf members. You can pick which column to import when you define mappings.

   **Parent-Child**
   Imports two columns of data with a parent-child relationship. For example, New England is the parent and Vermont is the child. Parent-Child is the default setting. You can specify the parent-child relationship in the source columns when you define mappings.

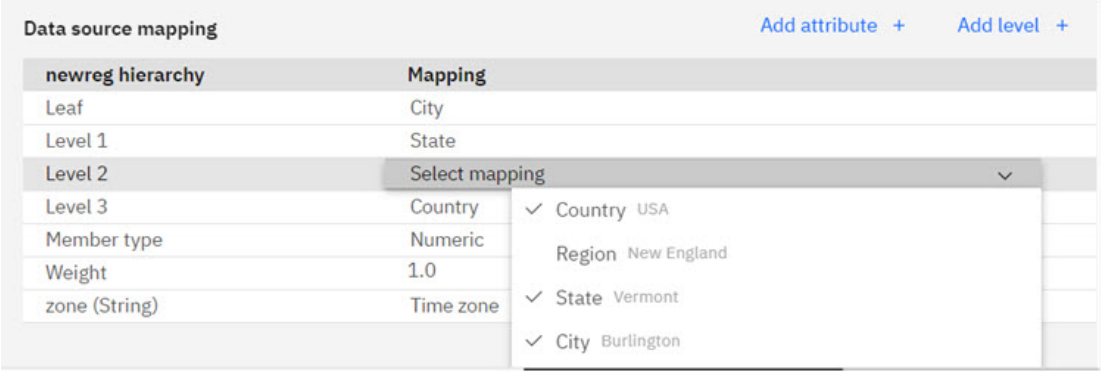
   **Multi-level**
   Creates more than two levels of consolidation in a dimension. You can add levels of consolidation and specify the consolidation path in the source columns when you define mappings.
8. If you chose to update an existing hierarchy during the import, you must specify how to update the hierarchy.

   Select **Recreate hierarchy** to delete all members from the existing hierarchy and recreate the hierarchy with the members that are contained in the import file.

   Select **Update hierarchy with new members** to update the existing hierarchy by adding any unique members from the import file to the hierarchy.
9. You can change the default **Decimal separator** and **Thousand separator**.
10. To create a process that contains the options that you specified, select the **Save as process** option and name the process.

    The process is saved in the **Processes** node of the database.
11. To schedule the process that you created, select the **Schedule process as** option and name the chore.

    A chore is created and set to run daily using the time of the initial import. The chore is saved in the **Chores** node of the database.
12. Map your data source columns to the hierarchy levels, leaf member weights, and (if applicable) attributes for your hierarchy.




a) Click the items in the **Mapping** column and map to the corresponding level of consolidation in your hierarchy. If there aren't enough levels in the hierarchy to accommodate mapping from the source, click **Add level** to insert a new higher level of consolidation.

b) To set the **Member type**, click the **Mapping** column and select either **Numeric** or **String** if all members have the same type, or select the column in the source file that contains the member type.

   Strings are used in pick lists and in cells where users type comments.

c) To assign the same weight to all leaf members in the hierarchy, type the weight value, such as 1.0 in the **Weight** field of the **Mapping** column. To import weights from the source file, select the column that contains the weight values.

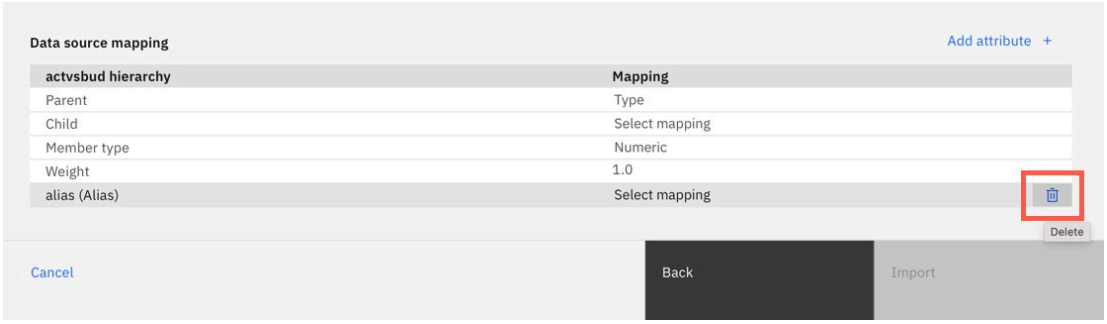
   To find out more about weights, see "Weights" on page 750.

d) To map an existing attribute to a column of data in the source, click the appropriate attribute field on the **Mapping** column and select the attribute.

e) To create a new attribute, click **Add attribute +**, enter the name of the attribute, and select the type: **String**, **Number**, or **Alias**.

   Then, click the corresponding field on the **Mapping** column to map a column in the source to the attribute.

   To remove an attribute that you added, hover over the attribute and click **Delete**.




13. Click **Import**.

## Dimension import file formats

When you import a file into a dimension, the file must be structured and formatted in a certain way to get the results that you want.

### File formats

The import data must be saved in a format, such as comma-separated (.csv) or with any other standard delimiter such as tab, space, semicolon, colon, vertical line, or caret (^).

**Note:** Files that contain a Byte Order Marker (BOM) might be corrupted when the file is imported.

## Parent-child dimension structure

If you want to create a dimension with a parent-child structure, the import file must have two columns, and the levels are defined in a parent-child relationship. Any extra columns are ignored. Headings are optional. A member can be both a child and a parent; the members Wales, England, Scotland, and Northern Ireland are both children and parents. An example of a dimension that was created by importing a file that is structured to have parent-child relationships is shown in the following picture.




The example can be created with a .csv file that contains either of the following formats.

Column-separated:

The first column contains the child dimension members; the second column contains the parent dimension members.

| Column 1 | Column 2 |
|---|---|
| Belfast | Northern Ireland |
| Birmingham | England |
| Bristol | England |
| Cardiff | Wales |
| Dundee | Scotland |
| Glasgow | Scotland |
| York | England |
| Inverness | Scotland |
| London | England |
| Manchester | England |
| Newcastle | England |
| Swansea | Wales |

| Column 1 | Column 2 |
|---|---|
| Wales | Great Britain |
| England | Great Britain |
| Scotland | Great Britain |
| Northern Ireland | Great Britain |

Comma-separated:

The value preceding the comma contains the child dimension members and the value after the comma contains the parent dimension members.

```
Belfast, Northern Ireland
Birmingham, England
Bristol, England
Cardiff, Wales
Dundee, Scotland
Glasgow, Scotland
York, England
Inverness, Scotland
London, England
Manchester, England
Newcastle, England
Swansea, Wales
Wales, Great Britain
England, Great Britain
Scotland, Great Britain
Northern Ireland, Great Britain
```

**Tip:** Some text editing applications may add additional metadata to your .csv file, which can affect the import process. For the best results, use a simple text editor for data that's comma-separated.

## Importing attributes, weights, and member type

You can import attributes, weights, and member types alongside members. Your file should have a column of data for the attribute, weight, and member type. Weight can only be imported for parent-child dimension structures.

The following example has 4 columns of data, with Family as the Parent, Model as the child, Engine Size as attribute, and Weight. You can import multiple attributes, and you can import into existing attributes, or you can create a new attribute during the file import.

```
Type,Model,Engine Size, Weight
Budget,L Series 1.8 L Sedan,1.8,1
Budget,L Series 2.0 L Sedan,2,1
Budget,L Series 2.5 L Sedan,2.5,1
Budget,S Series 1.8 L Sedan,1.8,1
Family,L Series 1.8 L Wagon,1.8,1
Family,L Series 1.8 L Wagon 4WD,1.8,1
Family,L Series 2.0 L Wagon,2,1
Family,L Series 2.0 L Wagon 4WD,2,1
Family,S Series 3.0 L Wagon 4WD,3,1
```

The following example has 2 columns of data. The first column is a leaf member, and the second column is the member type. The member type can be string (s), or number (n).

```
Name/Desc,s
Job Type,s
Job Code,s
FTE,n
50P Salary,n
Current Salary,n
Merit Pd,s
```

### Member names

The member names that are contained in the source files should follow “Naming conventions” on page 1264.

If the source file has multiple occurrences of a member name, the member is created by using the first occurrence of the member name.

You can have both mixed-case characters and spaces in member names, but case and spaces are ignored when member names are stored. This means that `North America`, `NorthAmerica`, and `north america` are equivalent member names in a database. If you dropped the following file onto the dimension editor, you see the member `North America` in your hierarchy, as that is the first occurrence that is encountered during file processing.



| | A |
|---|---|
| 1 | North America |
| 2 | NorthAmerica |
| 3 | north america |

### File size

No file size limit applies when you drop text files into the dimension editor, but large files might cause your web browser to timeout while you wait for a response. The text file is still processed and the members are inserted into the dimension hierarchy.

## Weights

The weight property controls how a child member rolls up to its immediate parent, whether that child is also a parent of another consolidation or a leaf member. Top level consolidations do not have a weighting.

Weight only applies to numeric members. The default weight for a member is 1, this gives a member a positive value. Weight can be used to change a positive value to a negative value, often -1. For example, if the unit price for a product is EUR 50 and the discount is EUR 5, apply a weight of -1 to the discount member. Applying a weight keeps the addition result logical.

You can also weight members as zero to keep them out of a total. For example, say you have a consolidation member called Drivers, and this is made up of Floorspace, Headcount, and Population growth. You would weight Floorspace, Headcount, and Population growth as zero because it makes no sense to add them together.

#### Gross margin example

You have an Accounts dimension with the following members: Gross margin, Sales, and Variable costs. The Gross margin member is calculated as Sales - Variable costs. In the dimension, it would be structured like this:

- Gross profit
  - Sales
  - Variable costs

You would give Sales a weight of 1 and Variable costs a weight of -1.

# Format dimension members in Planning Analytics Workspace

You can set the format for values in dimension members in the dimension editor. The default format is General.
For example, you can format a member as a date picker, as a currency value, or as scientific notation. You can also create custom formats.

**Note:**

Analysts or Modelers can set formats in a cube view or on an individual cell. Formats set directly in a view or cell override formats set on dimension members.

### Procedure

1. In the **Databases** tree, click **Dimensions**, right-click the dimension that you want to edit and click **Edit dimension**.
2. Click the **Attributes** tab.
3. Select the member or members for which you want to assign a format .
4. Click the action icon on the toolbar, then click **Format values**.
5. Select the format.

   If not already present, the first time you specify a format for a member a new Format attribute is added to the dimension.
6. You can set the format for all members in the dimension at the same time in the **Format** attributes column. Click the **Format** attributes header, then click the **Action** icon on the toolbar and click **Format values**.

### What to do next

To format a member as text, set the member type to string. For more information, see "Change the type of members to numeric or text in Planning Analytics Workspace" on page 753.

## Member formats

You can format dimension members in the dimension editor.

To apply the same format to all members of the dimension:

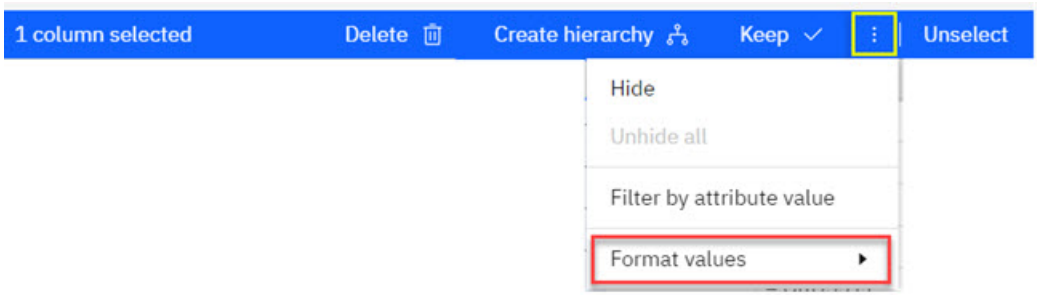
1. Open the dimension in the dimension editor.
2. Click the **Attributes** tab.
3. Right-click the **Format** column header.
4. Click **Format values**.
5. Click the format you want to apply to all the dimension members.

You can also apply a format to individual dimension members:

1. Open the dimension in the dimension editor.
2. Select one or more members to which you want to apply a format.

   To select multiple adjacent members, click the first member, then SHIFT+click the last member.

   To select multiple non-adjacent members, CTRL+click each member.
3. Click the **Action** icon on the edit dimension bar, then click **Format values**.




4. Click the format you want to apply to the selected dimension members.

You can choose from the following formats.

**Note:** The examples here use the actual value of -1234.567. The number changes based on the format used.

**General**
Displays numbers without commas to separate digits to the left of the decimal point. Negative values are prefixed with a minus sign (-).

`-1234.57`

**Fixed**
Displays numbers without commas to separate digits to the left of the decimal point. Negative values are surrounded by parentheses.

`(1234.57)`

**Comma**
Commas separate every third digit to the left of the decimal point.

`(1,234.57)`

**Rounded**
Commas separate every third digit to the left of the decimal point, rounded up to the nearest whole number. You can see the actual value by right-clicking in the cell.

`(12,346)`

**Percentage**
Multiplies numbers by 100 and displays a following percent sign (%). Digits to the left of the decimal point do not use commas, and negative values are prefixed with a minus sign (-).

`-123456.70%`

**Scientific**
Displays numbers in scientific notation. Scientific notation is a way of expressing large or small numbers. For example, the number 123,000,000,000 can be written as 1.23E+11.

Negative values are prefixed with a minus sign (-).

`-1.2E+3`

**Accounting**
Displays numbers with currency symbols and decimal points in a column. Negative values are surrounded by parentheses.

`$(1,234.57)`

**Currency**
Displays numbers with the currency symbol that is specified for your computer. Commas separate every third digit to the left of the decimal point. Negative values are surrounded by parentheses.

`($1,234.57)`

**Currency (rounded)**
Displays numbers with the currency symbol that is specified for your computer. Commas separate every third digit to the left of the decimal point, rounded up to the nearest whole number. You can see the actual value by right-clicking in the cell. Negative values are surrounded by parentheses.

`($1,235)`

**Date**
Gives you a date picker, and displays dates in a predefined format: `mm/dd/yyyy`.

`01/23/1989`

**Time**
Displays time in a predefined format: `hh:mm:ss`.

`12:30:00`

You can also define custom formats.

### Format examples

The following list shows the standard formats that are used in the dimension editor.

**Note:** d: means that ICU formatting is used. ICU is the standard format for IBM Planning Analytics Workspace.

**Fixed**
`d:0.00;(0.00)`

**Comma**
`d:#,##0.00;(#,##0.00)`

**Percentage**
`d:#.00%`

**Scientific**
`d:0.0E+0`

**Accounting**
`d:'$'#,##0.00;'$'(#,###.00)`

**Currency**
`d:'$'#,##0.00;('$'#,###.00)`

**Currency (rounded)**
`d:'$'#,##0;('$'#,###)`

**Date**
`c:MM/dd/yyyy`

**Time**
`d:H:mm:ssICD`

You can create custom formats by modifying these examples. To find out more about the syntax, see "Custom character formats in Planning Analytics Workspace" on page 754, "Numeric value formats" on page 757, and "Date and time display formats" on page 756.

## Change the type of members to numeric or text in Planning Analytics Workspace

You can change the properties of a member to make it consolidated, numeric, or a string (text). You can also change the weighting of a member.

**Note:** You can change the member type only for leaf members.

Make sure that all members that are strings in a cube are in the same dimension, along with any pick list members. The dimension that contains string members and pick list members must be the last dimension in the cube.

### Procedure

1. On the **Databases** tree, right-click the dimension you want to work on and click **Edit dimension**.
2. Click the **Members** tab on the dimension editor.
3. To change the type of a member, select the member name, then select the desired member type in the **Type** column.

   **Note:** Dimensions that contain members that have the String type, or are pick lists, must be last in a cube. String type members and pick list members must be in the same dimension.
4. To change the weight of a member, select the member name, then enter the desired weight in the **Weight** column.

   Weight can be used to change a positive value to a negative value, often -1.

For example, if the unit price for a product is EUR 50 and the discount is EUR 5, apply a weight of -1 to the discount member. Applying a weight keeps the addition result logical. To find out more, see “Weights” on page 750.

## Custom character formats in Planning Analytics Workspace

You can create a custom format in the dimension editor to display characters.

To display text, change the member properties to a String type.

1. In the dimension editor, select the dimension member for which you want to create a custom format.
2. Click the **Action** icon on the toolbar, then click **Format values**, **Custom format**.
3. Type your custom format.
4. Click **OK**.

In IBM Planning Analytics Workspace, you must start the custom format with c:

Format expressions for characters can have one section, or two sections separated by a semicolon (;). If you use one section, the format applies to all string data that can occur in the cell. If you use two sections, the first section applies to string data, and the second section applies to null values and zero-length strings.

For example:

```
c:<@@@;"No Value"
```

This format displays three lower-case characters if the cell contains string data, or the string No Value if the cell contains a null value or a zero-length string.

**Note:** Dimensions that contain members are formatted as strings, or are pick list, must be last in a cube. String type members and pick list members must be in the same dimension.

The following table describes how to construct a custom format string for a string element:

| Format String Character | Description |
|---|---|
| @ | Character placeholder. If the string has a character in the position where the at symbol (@) appears, that character displays. If no character appears in that location, a space displays.<br>Example:<br>Suppose a cell contains the following string:<br>The quick brown fox<br>If you apply this format string:<br>`c:@@@@@`<br>Displays:<br>n fox<br>**Note:** Placeholders are populated from right to left unless you enter an exclamation point (!) character in the format string. |

| Format String Character | Description |
|---|---|
| & | Character placeholder. If the string has a character in the position where the ampersand symbol (&) appears, that character displays. In this case, a space is considered a character and will be displayed. If no character appears in that location, nothing displays.<br>Example:<br>Suppose a cell contains the following string:<br>The quick brown fox<br>If you apply this format string:<br>`c:&&&&&`<br>Displays:<br>nfox<br>**Note:** Placeholders are populated from right to left unless you enter an exclamation point (!) character in the format string. |
| < | Displays all characters in lowercase. < must be used in conjunction with either @ or &, for example:<br>`c:<&&&&&` |
| > | Displays all characters in uppercase. > must be used in conjunction with either @ or &.<br>Example:<br>Suppose a cell contains the following string:<br>The quick brown fox<br>If you apply this format string:<br>`c:>@@@@@`<br>Displays:<br>N FOX |
| ! | Forces placeholders to fill from left to right. ! must be used in conjunction with either @ &.<br>Example:<br>Suppose a cell contains the following string:<br>The quick brown fox<br>If you apply this format string:<br>`c:!>@@@@@`<br>Displays:<br>THE QU |

## Date and time display formats

The following table lists characters that can appear in a format string for date and time formats.

| Format String Character | Description |
|---|---|
| : | Time separator. (In some locales, other characters may be used to represent the time separator.)<br>This character separates hours, minutes and seconds when time values are formatted. The actual character used as the time separator in formatted output is determined by your system settings. |
| / | Date separator. (In some locales, other characters may be used to represent the date separator.)<br>The date separator separates the day, month, and year when date values are formatted. The actual character used as the date separator in formatted output is determined by your system settings. |
| C | Displays the date as ddddd and displays the time as ttttt, in that order. Displays only date information if there is no fractional part to the date serial number. Displays only time information if there is no integer portion.<br>Example: 10/10/98 05:12:12 |
| d | Displays the day as a number without a leading zero (1-31). |
| dd | Displays the day as a number with a leading zero (01-31). |
| ddd | Displays the day as an abbreviation (Sun-Sat). |
| dddd | Displays the day as a full name (Sunday-Saturday). |
| ddddd | Displays the date as a complete date (including day, month, and year), formatted according to the long date setting recognized by your system. For Microsoft Windows, the default long date is m/d/yy. |
| dddddd | Displays a date serial number as a complete date (including day, month, and year), formatted according to the long date setting recognized by your system. For Microsoft Windows, the default long date format is mmmm dd, yyyy. |
| w | Displays the day of the week as a number. (1 for Sunday through 7 for Saturday). |
| ww | Displays the weeks of the year as a number (1 - 54) |
| m | Displays the month as a number without a leading zero (1 - 12). If m immediately follows h or hh, the minute rather than the month displays. |
| mm | Displays the month as a number with a leading zero (01 - 12). If m immediately follows h or hh, the minute rather than the month displays. |
| mmm | Displays the month as an abbreviation (Jan - Dec). |
| mmmm | Displays the month as a full month name (January - December) |

| Format String Character | Description |
|---|---|
| q | Displays the quarter of the year as a number (1 - 4). |
| y | Displays the day of the year as a number ( 1 - 366). |
| yy | Displays the year as a two-digit number (00 - 99). |
| yyy | Displays the year as a four-digit number (0100 - 9999). |
| h | Displays the hour as a number without leading zeros (0 - 23). |
| hh | Displays the hour as a number with leading zeros (01 - 23). |
| n | Displays the minute as a number without leading zeros (0 - 59). |
| nn | Displays the minute as a number with leading zeros (00 - 59). |
| s | Displays the second as a number without leading zeros (0 - 59). |
| ss | Displays the second as a number with leading zeros (00 - 59). |
| t t t t t | Displays a time as a complete time (including hour, minute, and second), formatted using the system time separator. A leading zero displays if the time is before 10:00 AM or 10:00 PM. For Microsoft Windows, the default time format is hh:mm:ss. |
| AM/PM | Uses the 12-hour clock. Displays an uppercase AM with any hour before noon; displays an uppercase PM with any hour between noon and 11:59 P.M. |
| am/pm | Uses the 12-hour clock. Displays a lowercase AM with any hour before noon; displays a lowercase PM with any hour between noon and 11:59 P.M. |
| A/P | Uses the 12-hour clock. Displays an uppercase A with any hour before noon; displays an uppercase P with any hour between noon and 11:59 P.M. |
| a/p | Uses the 12-hour clock. Displays a lowercase a with any hour before noon; displays a lowercase p with any hour between noon and 11:59 P.M. |
| AMPM | Uses the 12-hour clock. Displays the AM string literal with any hour before noon; displays the PM string literal with any hour between noon and 11:59 P.M. AMPM can be either uppercase or lowercase, but the case of the string displayed matches the string as defined by your system settings. For Microsoft Windows, the default format is AM/PM. |

## Numeric value formats

Format expressions for numbers have up to four sections separated by semicolons (;). The number of sections determines which types of values are affected.

- If a format has one section, that section applies to all values.
- If a format has two sections, the first section applies to positive values and zeros, and the second section applies to negative values.
- If a format has three sections, the first section applies to positive values, the second section applies to negative values, and the third applies to zeros.

- If a format has four sections, the first section applies to positive values, the second section applies to negative values, the third applies to zeros, and the fourth applies to NULL values.

The following table describes how to construct a format string for numeric values:

| Format String Character | Description |
|---|---|
| # (number sign) | Digit placeholder.<br>The # placeholder displays only significant digits and does not display insignificant zeros. In the decimal value .90, the 0 is considered insignificant. The value would be displayed as .9 when the # placeholder is used.<br>If a number has more digits to the right of the decimal point than there are placeholders in the format string, the number rounds to as many decimal places as there are placeholders. If there are more digits to the left of the decimal point than there are placeholders, the extra digits are displayed. |
| | The following examples illustrate the use of the # placeholder.<br>Example:<br>Value: 123.896<br>Format String: #.##<br>Displays: 123.9<br>Example:<br>Value: 456.873<br>Format String: #.##<br>Displays: 456.87<br>Example:<br>Value: 34.5678<br>Format String: #.###<br>Displays: 34.568<br>You can combine the # and 0 placeholders in a format string. |

| Format String Character | Description |
|---|---|
| 0 (zero) | Digit placeholder.<br>The 0 placeholder displays insignificant zeros if a number has fewer digits than there are zeros in the format string.<br>If a number has more digits to the right of the decimal point than there are placeholders in the format string, the number rounds to as many decimal places as there are placeholders. If there are more digits to the left of the decimal point than there are placeholders, the extra digits are displayed.<br>The following examples illustrate the use of the 0 placeholder.<br>Example:<br>Value: 23.896<br>Format String: 0.00<br>Displays: 23.90<br>Example:<br>Value: 16.8<br>Format String: 0.000<br>Displays: 16.800<br>Example:<br>Value: 7.12<br>Format String: 000.0<br>Displays: 007.1<br>You can combine the # and 0 placeholders in a format string. |
| E- E+<br>e- e+ | Scientific format.<br>If the format string contains at least one digit placeholder (0 or #) to the right of E-, E+, e-, e+, the number displays in scientific format and E or e is placed between the number and its exponent.<br>The number of digit placeholders to the right determines the number of digits in the exponent. Use E- or e- to place a minus sign next to negative exponents. Use E+ or e+ to place a minus sign next to negative exponents and a plus sign next to positive exponents. |
| - + $ () | Displays a literal character. To display a character other than one of those listed, precede it with a backslash (\) or enclose it in double quotation marks. (" ").<br>Numeric Value: -1000.00<br>Format String: ($-#.##)<br>Displays: ($-1000.00)<br>The following characters cannot be displayed as literals: a, c, d, h, m, n, p, q, s, t, w, y, /, :, #, 0, %, E, e, comma(,), period(.), @, &, <, >, and ! |

| Format String Character | Description |
|---|---|
| \ | Displays the next character in the format string.<br>Numeric Value: 100<br>Format String: `\t\o\t\a\l\=#`<br>Displays: total=100 |
| "ABC" | Displays the string inside the double quotes. (In this example, ABC would display.)<br>Numeric Value: 100<br>Format String: `#" units"`<br>Displays: 100 units |

# Add members to a dimension or hierarchy in Planning Analytics Workspace

You can add members to a dimension or hierarchy one at a time or by copying and pasting a number of members from an editor such as Microsoft Excel.

You can also import dimension members from a file. For more information, see Import members and attributes into a dimension.

This procedure describes how to add members individually in the dimension editor.

Before modifying a dimension, you might want to reserve the dimension to prevent other users from simultaneously modifying the dimension.

## Procedure

1. In the **Databases** tree, expand the database that contains the dimension that you want to edit.
2. Expand the **Dimensions** node, then right-click the dimension that you want to edit and click **Edit dimension**.
3. Select the hierarchy you want to modify.
4. Click the **Members** tab.
5. Select a member.

   Generally, you should select a member that is immediately adjacent to the new member you want to create. You can later choose the relationship or position of the new member to the selected member.
6. Click **Add member**, then click the desired option for the new member.
7. Type a name for the member.

   If you press ENTER after typing a member name, you can immediately begin entering a new member at the same level in the dimension.
8. Optionally, select a **Type** and **Weight** for the new member.

   By default, new leaf members are added as numeric members with a weight of 1.

   If you add a member as a parent, the member is always added as a consolidated member.
9. Click **Commit** to save your changes to the dimension or hierarchy.

   You can also quickly add a root level member to your dimension by clicking **Add root member**.

## What to do next

- Format the members. For example, you can format a member as a date or time picker, as a currency value, or as scientific notation. See “Member formats” on page 751.

- Create attributes for dimension members. Attributes provide extra information about dimension members, see “Member attributes” on page 771.
- Drag a dimension member as parent, child, or peer of other members. See “Drag members as children or peers in Planning Analytics Workspace” on page 761.

## Drag members as children or peers in Planning Analytics Workspace

You can drag a member as a child of another member or as a peer of another member to change the dimension or hierarchy structure.

### About this task

The following limitations apply:

- You cannot drag a member as a parent of another member.
- If you drag a member as a child of another member, a dialog box warns you about potential data loss. This warning appears only if the dimension that you are editing is part of a cube and this action converts a leaf level member to a consolidated member.

### Procedure

1. In the **Databases** tree, expand the database that contains the dimension that you want to edit.
2. Expand the **Dimensions** node, then right-click the dimension that you want to edit and click **Edit dimension**.
3. In the dimension editor, select the member that you want to drag.
4. Drag the member onto another member to make it a child of that member.

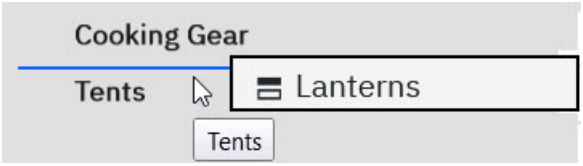
   When you drag a selected member onto another member, the other member has a blue border, which indicates that dropping the selected member will create a parent-child relationship. The icon next to the selected member also indicates that dropping onto the other member will create a parent-child relationship.

5. Drag the selected member above or below another member to drop it as a peer of that member.

   When you drag a selected member above or below another member, a blue line appears to indicate where you can drop the selected member as a peer. The icon next to the selected member also indicates that dropping will insert the selected member as a peer.

   In this example, the blue line and the icon indicate that dropping the selected member here inserts Lanterns above and as a peer of Tents.

   Note that depending on the fine positioning of the mouse pointer, the icon can change to indicate the relative placement of the selected member.

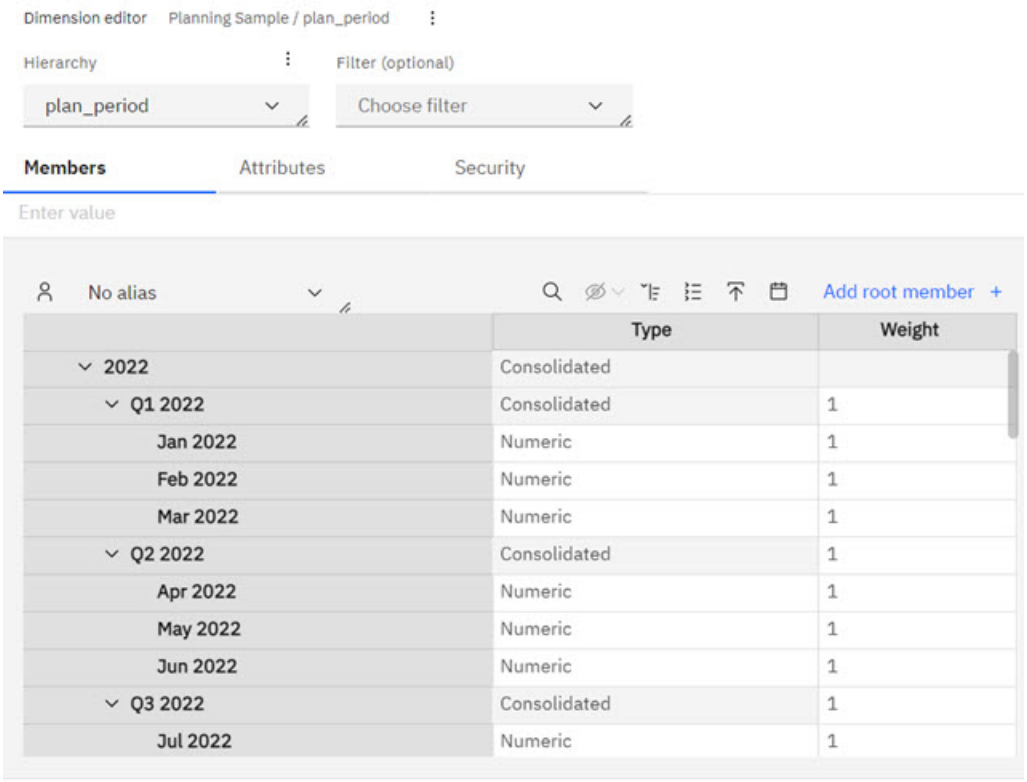
   In this example, the blue line appears between Cooking Gear and Tents, as in the previous example. But the mouse pointer is closer to Cooking Gear, so the icon indicates that dropping the selected member here inserts Lanterns below and as a peer of Cooking Gear.




# Time dimensions in Planning Analytics Workspace

A time dimension defines the time periods that are used in your model. Most models use a time dimension.

For example, here's a time dimension that spans multiple years and uses months as leaf members. Months consolidate to quarters, which consolidate to years.




| | Type | Weight |
|---|---|---|
| 2022 | Consolidated | |
| Q1 2022 | Consolidated | 1 |
| Jan 2022 | Numeric | 1 |
| Feb 2022 | Numeric | 1 |
| Mar 2022 | Numeric | 1 |
| Q2 2022 | Consolidated | 1 |
| Apr 2022 | Numeric | 1 |
| May 2022 | Numeric | 1 |
| Jun 2022 | Numeric | 1 |
| Q3 2022 | Consolidated | 1 |
| Jul 2022 | Numeric | 1 |

A time dimension has the following member attributes:

**Last Period**
The final time period in the sequence.

**First Period**
The initial time period in the sequence relative to the current time.

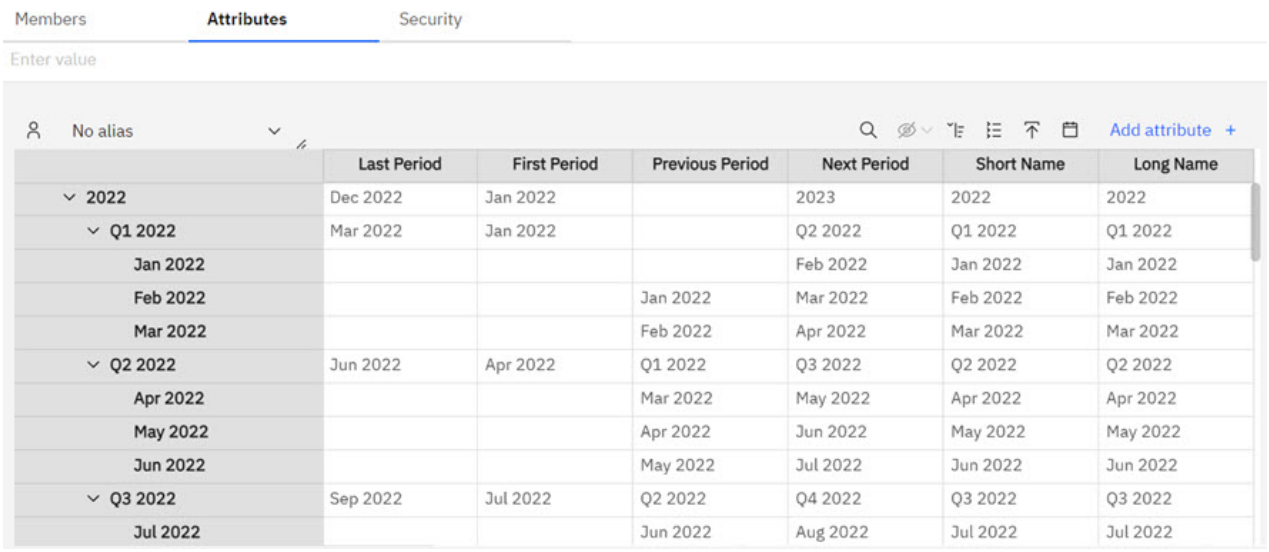
**Previous Period**
The previous time period in the sequence.

**Next Period**
The next time period in the sequence.

**Short Name**
An alias. The short name is populated with the Member ID (the member name) - for example Jan 2018. You can modify the short name by typing in the cell.

**Long Name**
An alias. The long name is populated with the Member ID. You can modify the long name by typing in the cell.

To view the time attributes, open the time dimension in the dimension editor and click the Attributes tab.




| | Last Period | First Period | Previous Period | Next Period | Short Name | Long Name |
|---|---|---|---|---|---|---|
| 2022 | Dec 2022 | Jan 2022 | | 2023 | 2022 | 2022 |
| Q1 2022 | Mar 2022 | Jan 2022 | | Q2 2022 | Q1 2022 | Q1 2022 |
| Jan 2022 | | | | Feb 2022 | Jan 2022 | Jan 2022 |
| Feb 2022 | | | Jan 2022 | Mar 2022 | Feb 2022 | Feb 2022 |
| Mar 2022 | | | Feb 2022 | Apr 2022 | Mar 2022 | Mar 2022 |
| Q2 2022 | Jun 2022 | Apr 2022 | Q1 2022 | Q3 2022 | Q2 2022 | Q2 2022 |
| Apr 2022 | | | Mar 2022 | May 2022 | Apr 2022 | Apr 2022 |
| May 2022 | | | Apr 2022 | Jun 2022 | May 2022 | May 2022 |
| Jun 2022 | | | May 2022 | Jul 2022 | Jun 2022 | Jun 2022 |
| Q3 2022 | Sep 2022 | Jul 2022 | Q2 2022 | Q4 2022 | Q3 2022 | Q3 2022 |
| Jul 2022 | | | Jun 2022 | Aug 2022 | Jul 2022 | Jul 2022 |

You can add attributes, for example, you might want to add translations for the text. To find out more, see "Create and edit member attributes in Planning Analytics Workspace" on page 771.

## Create a time dimension in Planning Analytics Workspace

You can create a time dimension directly from the **Databases** tree or from an empty dimension editor.

### Procedure

1. Right-click the **Dimensions** node in your database on the **Databases** tree, then click **Create time dimension**.

   Alternatively, you can create a time dimension by clicking the **Configure time dimension** option when you create a new dimension. The **Configure time dimension** option is only available for a dimension with one hierarchy and no members.
2. If required, enter a name for the dimension.
3. Select the **Start year** for the dimension.
4. Select the **End year** for the dimension.
5. Answer the **How frequently do you plan?** question by selecting the granularity of the dimension.

   **Months**
   This option gives you three levels: Months, Quarters, Years.

   **Quarters**
   This option gives you two levels: Quarters, Years.

   **Years**
   This option gives you one level: Years.
6. Check the preview to confirm that it matches your expectation, then click **Save**.

   You can modify the time dimension in the dimension editor after you create it.

## Extend a time dimension in Planning Analytics Workspace

You might find it necessary to extend the period covered by your time dimension.

### Procedure

1. Open the time dimension in the dimension editor.
2. Click the **Configure time** icon.
3. Select a new **End year**.

4. Click **Extend**.

### Results

The dimension is extended to the year you select, using the granularity (months, quarters, years) you specified when you initially created the time dimension.

## Reserve and release a dimension in Planning Analytics Workspace

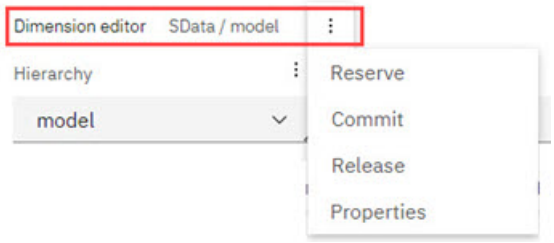
When you work in the dimension editor without reserving a dimension, any changes you make are instantly saved to the database so that anyone who has access to the database can see the updated dimension. However, if the dimension is not reserved, other users can simultaneously edit the dimension.

You can reserve a dimension to prevent other people from editing the dimension until your changes are complete. When you reserve a dimension, any changes that you make are not saved to the database until you click Commit on the action menu.

You must have at least Write privilege to a dimension to reserve the dimension.

### Procedure

1. In the dimension editor, click the action icon ⋮ next to the dimension name, then click **Reserve**.




When a dimension is reserved, an icon is displayed next to the dimension name.




Any user who opens the dimension in the dimension editor can hover over the icon to see which user reserved the dimension.

You can edit only the dimension members and hierarchies when a dimension is reserved; you can't edit attributes or member security.

2. After you complete your modifications to the dimension, click **Commit** on the dimension edit bar to save your changes.




**Note:** Because the dimension is reserved, this **Commit** action does not save your changes to the database. Rather, the changes are saved to a temporary 'sandbox.' Changes are not saved to the database until you click **Commit** on the action menu, as described in Step 4.

3. To undo changes to the dimension before the dimension is saved to the database, click the action icon ⋮, then click **Release**. This action reverts any changes you've made since you reserved the dimension, even changes that were committed after you click **Commit** on the dimension action bar.
4. When your changes are complete, click the action icon ⋮, then click **Commit** to release the dimension reservation and save your changes to the database.

**Related tasks**

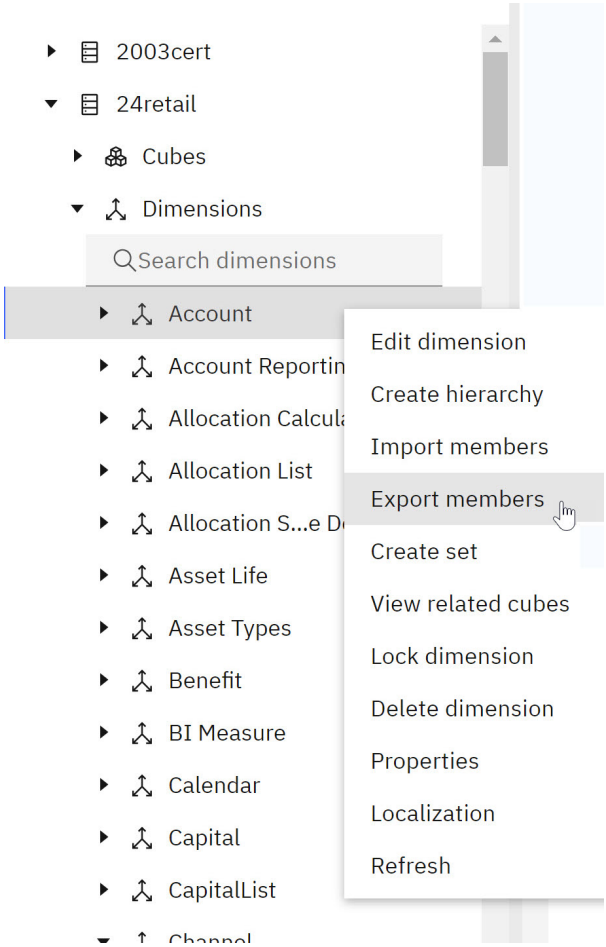
"Lock and unlock a dimension in Planning Analytics Workspace" on page 766

You can lock a dimension so that only users who have Admin privileges for the dimension can modify the dimension.

## Export a dimension or hierarchy from a modeling workbench

You can export a dimension or hierarchy to a `.csv` file. The exported file is saved to your local `Downloads` directory.

To export, right-click the dimension or hierarchy on the modeling workbench **Databases** tree, then click **Export members**.




You can also click the **Export members** icon on the dimension editor to export a dimension.

When you export a dimension or hierarchy, the resulting `.csv` file includes the following information about each member:

- name
- parent/child relationship
- type
- weight
- any defined attribute names and values

  Attribute names are prefixed with either ATTR_N or ATTR_S when they are exported, depending on the attribute type (numeric or string, respectively).

# Lock and unlock a dimension in Planning Analytics Workspace

You can lock a dimension so that only users who have Admin privileges for the dimension can modify the dimension.

## About this task

A user must have either Admin or Lock privilege for a dimension to lock the dimension.

When a user locks a dimension, only those users who have Admin rights for that dimension can modify the dimension. Even the user who locks the dimension cannot modify it, unless they have Admin rights for that dimension.

When a dimension is locked, options are disabled for these actions:

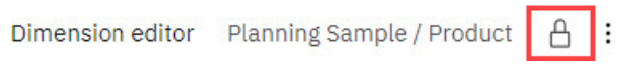
- Add members to any of the hierarchies in the dimension
- Remove members from any of the hierarchies in the dimension
- Create a hierarchy in the dimension
- Delete a hierarchy in the dimension

Other users with at least Read privilege to the dimension can create, delete, or update subsets under the locked dimension.

## Procedure

1. To lock a dimension, right-click the dimension on the **Databases** tree in either a book or a modeling workbench, then click **Lock dimension**. A lock icon on the dimension name indicates that the dimension is locked.

   A lock icon on the header of the new experience dimension editor also indicates that the dimension is locked.

   If a dimension is locked while it is open in the new experience dimension editor, you must refresh the dimension editor to see the lock icon.
2. To unlock a dimension, right-click the dimension on the **Databases** tree in either a book or a modeling workbench, then click **Unlock dimension**.

**Related tasks**

"Reserve and release a dimension in Planning Analytics Workspace" on page 764

# Lock and unlock an element in IBM Planning Analytics Workspace

You can lock and unlock an element in IBM Planning Analytics Workspace.

When a user locks an element, only those users who have lock permissions or higher can unlock the element and update the data that it identifies. An admin user can unlock the element, but they cannot update the data. Even the user who locks the element cannot update its data, unless they have Admin rights for that element. An admin user cannot update the data in the .

Consider locking an element as a way to permanently archive the data that it identifies. Locks stays in place after the remote server shuts down.

**Note:** To use this feature, IBM Planning Analytics 2.0.9.19/TM1 11.8.023 is required.

## Locking an element

To lock an element, complete the following:

1. Right-click the element you want to lock in the **Dimension editor**.

2. Click **Lock**.

   The lock icon appears beside the locked element.

### Unlocking an element

To unlock an element:

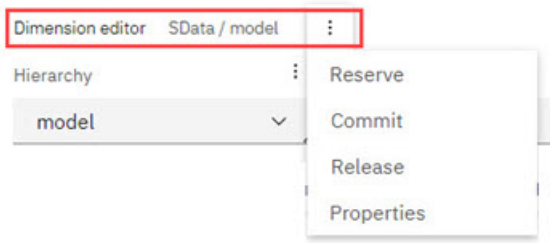
1. Right-click on a locked element in the **Dimension editor**.
2. Click **Unlock**.

   The lock icon disappears from beside the locked element.

## Manage dimension properties

You can view and edit dimension properties in the dimension editor.

Click the action icon next to the dimension name, then click **Properties**.




Some of these properties cannot be edited, they are updated by the system.

| Property name | Description |
|---|---|
| ALLLEAVESHIERARCHYNAME | This property stores the name given to the all leaves hierarchy. The all leaves hierarchy is present in any dimension that contains more than one hierarchy. |
| DEFAULT_HIERARCHY | Not currently used. |
| DEMANDLOAD | Not currently used. |
| LAST_TIME_UPDATED | Time (GMT) at which a dimension was last updated. The time format is yyyymmddhhss. |
| LASTREPSYNCCHANGEDTIME | |
| LOCK | Indicated when a dimension is locked. |
| REPLICATION | For replicated cubes, this property stores the name of the replication connection with which the replicated cube is associated. |
| REPSRCNAME | For replicated dimensions, this property stores the name of the source dimension. |
| REPSTATUS | If a dimension is replicated on the server, the value of REPSTATUS is Copied. Otherwise the REPSTATUS value is empty. |

| Property name | Description |
|---|---|
| SORTCOMPONENTSSENSE | When the immediate components (children) of a consolidation are sorted, this property stores the sense applied to the sorting. Components can be sorted in either the ASCENDING or DESCENDING sense. |
| SORTCOMPONENTSTYPE | This property indicates the type of sorting that is applied to the immediate components (children) of a consolidation. There are two types of sorting: BYNAME and BYINPUT. The BYINPUT sort does not actually enforce any sorting, it leaves the components in the order they appeared the last time the dimension was saved. |
| SORTELEMENTSSENSE | When dimension elements are sorted automatically, this property stores the sense applied to the sorting. Elements can be sorted in either the ASCENDING or DESCENDING sense. |
| SORTELEMENTSTYPE | When dimension elements are sorted automatically, this property stores the type of sorting used. There are four types of element sorting: BYNAME, BYLEVEL, BYHIERARCHY, and BYINPUT. The BYINPUT sort does not actually enforce any sorting, it leaves the elements in the order they appeared the last time the dimension was saved. If a sorting is not specified, BYINPUT applies.<br>SORTELEMENTSTYPE applies to all dimension elements, both consolidations and leaf elements. |
| SYNCATTRIBUTES | For replicated cubes, this property indicates whether element attributes are synchronized when the associated dimension is synchronized. |
| SYNCSUBSETS | For replicated cubes, this property indicates whether subsets are synchronized when the associated dimension is synchronized. |
| USESEPARATECONTROLCUBES | Not currently used. |
| VISIBILITY | |

# Manage dimension attributes

Dimension attributes provide information about a dimension.

## About this task

You can create, view, and edit the values for dimension attributes, but you cannot delete attributes in the editor. Attributes for dimensions can be created in a TurboIntegrator process that uses the available dimension attribute functions.

**Important:** You cannot delete attributes even if you have administrative access on a dimension or an attribute cube. To delete an attribute, you must be a member of the ADMIN group.

## Procedure

1. On the **Database** tree on a modeling workbench, expand a database and right-click the **Dimensions** node.
2. Click **Dimension Attributes**.

A modified view of the }DimensionAttributes control cube opens, with individual dimensions on the row axis and attributes on the column axis.

3. Click a cell at the intersection of a dimension and an attribute, then enter or modify the attribute value.

   You can use cut, copy, and paste to manage attribute values. You can also use the standard view options to manage sets and to select multiple cells, rows, or columns in the editor.

# Hierarchies

You can use hierarchies in dimensions to see alternative rollups of the data in the same view. For example, you might have a hierarchy that shows an organization that is rolled up by region, and a hierarchy that shows an organization that is rolled up by vice-president.

You can also create hierarchies of member attributes such as color, size, and type. Use these hierarchies for filtering and analysis.

All dimensions include a single hierarchy by default.

Here are some of the benefits of using hierarchies:

- Hierarchies can improve query performance.
- You can turn attributes into hierarchies. Modeling attributes as hierarchies instead of dimensions can save memory space. If you use hierarchies, you can have cubes with fewer dimensions. Hierarchies act as virtual dimensions.
- Hierarchies give you greater flexibility. A single dimension can contain multiple hierarchies; hierarchies act as virtual dimensions, and you can display multiple hierarchies in the same view.

**Note:** Creating multiple hierarchies in a dimension is disabled in Planning Analytics Workspace by default. You must set the `EnableNewHierarchyCreation` configuration parameter in the `tm1s.cfg` file to `true` to enable hierarchy creation in Planning Analytics Workspace. For more information, see Parameters in the tm1s.cfg File in the *Planning Analytics Installation and Configuration* documentation.

## Sort order

The sort order for hierarchies is determined on the following basis:

- The default hierarchy (typically, the hierarchy with the same name as the dimension) appears first.
- The all-leaves hierarchy (typically called Leaves) appears last.
- All other hierarchies are sorted alphabetically by their caption.

# Work with hierarchies in Planning Analytics Workspace

You can create a hierarchy in the dimension editor when you create a dimension or you can add a hierarchy to an existing dimension.

You can also create a hierarchy by saving an existing hierarchy with a new name and you can create a hierarchy from a string attribute.

## Procedure

1. To create a hierarchy in the dimension editor, click the action icon ⋮ next to **Hierarchy**, then click **Create hierarchy**.
2. To create a hierarchy from the **Databases** tree:
   a) Right-click a dimension and click **Create hierarchy**.

      By default, the hierarchy is created on the same database and in the same dimension from which you clicked **Create hierarchy**, but you can use the **Database** and **Dimension** selectors to create a hierarchy on a different database and within a different dimension.

   b) Enter the name of the new hierarchy and click **Create**.

c) Add members to the hierarchy using any of the methods described in “Create a dimension in Planning Analytics Workspace” on page 744.

3. To create a hierarchy from an existing hierarchy in the dimension editor:
   a) Open the existing hierarchy in the dimension editor.
   b) Click the action icon ⋮ next to **Hierarchy** and click **Save hierarchy as**.
   c) Name the hierarchy and click **Save as**.
   The members and attributes of the original hierarchy are copied.
4. To create a hierarchy from a string attribute, click the attribute name on the **Attributes** tab in the dimension editor, then click **Create hierarchy**. A new hierarchy is created with the same name as the attribute.
5. To edit an existing hierarchy, open the parent dimension of the hierarchy in the dimension editor, then click the **Hierarchy** selector and click the hierarchy you want to edit. Alternatively, right-click the hierarchy in the **Databases** tree, then click **Edit hierarchy**.
6. To delete a hierarchy, right-click the hierarchy in the **Databases** tree, then click **Delete hierarchy**.

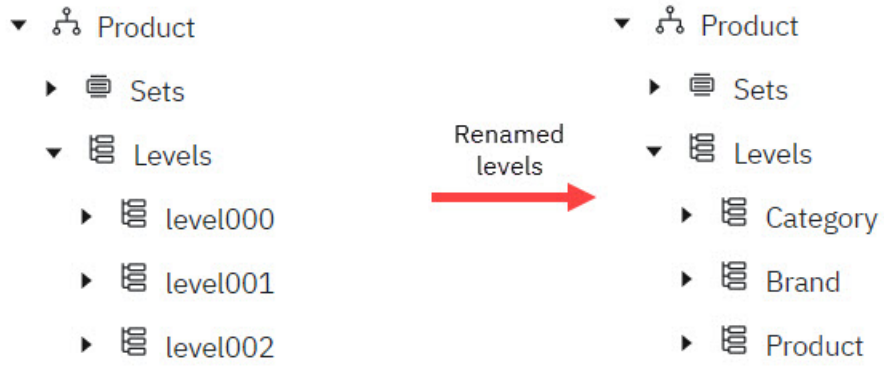
## Rename hierarchy levels in the dimension editor

Rename hierarchy levels directly in the new experience dimension editor.

### About this task

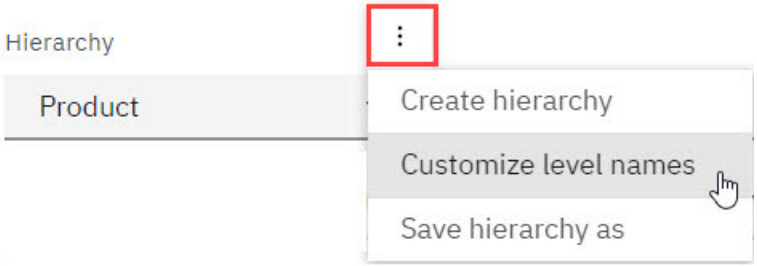
By default, levels within a hierarchy are named `level000` (highest level of consolidation0, `level001`, `level002`, and so on.

You can assign custom level names in the dimension editor to make hierarchy levels more easily identifiable. For example, the Product hierarchy has three levels corresponding to Category, Brand, and Product. Customizing the levels with the relevant names makes it easier to work in the hierarchy.




### Procedure

1. Open the hierarchy in the dimension editor.
2. Click the **Action** button next to the Hierarchy label, then click **Customize level names**.




3. Enter a custom name for each hierarchy level.

4. Click **Done**.

## Set a default member for a hierarchy

Specify a default member in a hierarchy to use when no specific member is referenced.

### About this task

Use the dimension editor to set a default member for any hierarchy. The default member is automatically used when no specific member is referenced in, for example, MDX queries that do not explicitly include the hierarchy. Specifying a default member helps you to obtain consister query results and simplifies report development.

### Procedure

1. Open the hierarchy in the dimension editor.
2. Right-click the member that you intend to set as the default member.
3. Select **Set as default**.

### Results

A star icon indicates the default member in a hierarchy.

# Member attributes

Member attributes help to explain or describe a member in a dimension.

For example, suppose that you have a dimension that lists cars. You could have attributes that describe fuel type, engine size or type, and number of doors.

You can show attributes in a table and you can select which attributes you want to show. You can sort on attributes, and you can convert attributes into a hierarchy. Converting attributes into hierarchies is useful if you want to group dimension members by attribute. For example, suppose that you want to display cars that have a gasoline engine, you can add an attribute called Engine type, then create a hierarchy from the Engine type attribute. You would then be able to group members by Engine type attribute in the view.

You add member attributes in the dimension editor.

## Create and edit member attributes in Planning Analytics Workspace

Attributes for dimension members provide additional information and context about members, which can help users to understand the information in reports and views. You can create attributes in the dimension editor.

### Procedure

1. Open the dimension for which you want to create or edit attributes in the dimension editor.
2. Click the **Attributes** tab.
3. Click **Add attribute**.
4. Enter the attribute name and select the attribute type.

   A column is created for the new attribute, with the attribute name in the header.
5. Type or paste attribute values into the cells.

   You can view an icon indicating the attribute type in column headers. Click  and enable the **Attribute type** option to see the attribute type on column headers. Disable this option to hide the attribute type on column headers.

### *Captions for members*

A caption is an attribute that has the alias attribute type, and a name of `Caption`.
You can choose whether to display the member caption, the member ID, or an alias in the set editor. If a caption exists, the caption is the default display name.

Captions can also be created for other objects such as cubes, and dimensions.

### Caption

In IBM Planning Analytics Workspace, the value for caption must be unique. The caption is the default value.

### Member ID

The member ID is the name that a member was given when it was first created. The member ID cannot change.

### Aliases

Aliases are attributes that have the attribute type of alias. Captions must have the alias type in Planning Analytics Workspace.

## Show member attributes in the cube view

Some dimension members have attributes. Attributes help to explain or describe a member in a dimension. You can show member attributes in a table.

For example, suppose you have a dimension with car models with an attribute called CustomerTarget, and you want to see the customer target for each car model in the table. You can choose to show or hide this attribute.

### Procedure

1. Right-click either the row selector or the column selector, then click **Show attributes**.
2. In the **Show attributes** pane, select the attributes that you want to show in the view.

   You can select several attributes and you can choose the order in which the attributes are displayed by moving them using the up and down arrows on the Selected pane.
3. Click **Apply** to save your choices and to return to the view.
4. You can sort the attributes in either ascending or descending order. Right-click either a row attribute selector or a column attribute selector, then click the desired sort order.
5. To hide attributes, right-click the row or column selector, then click **Show attributes** > **Hide attributes**.

# Pick lists

A pick list contains values that a user can select in a cell.

The values in a pick list correspond to the members of a dimension or set of a dimension. If the members of the dimension or the set change, the values available in the pick list also change. A pick list can also contain a static list of values that you specify when you create the pick list.

You can create the following types of pick lists:

**Static**
The pick list contains only the values that you specify.

**Subset**
The pick list references the values in the set. If the values in the set change, then the values in the pick list also change.

**Dimension**
The pick list references the values in the dimension. If the values in the dimension change, then the values in the pick list also change.

For example, a pick list is useful when staff managers do performance planning; they can assign a performance grade to their staff by using pick lists to select from a fixed list of values, such as **Low**, **Medium**, **High**, and **Excellent**, instead of typing a free form text string. If you set a pick list on a member with an existing format, the operation removes the format.

A pick list could also be used to select job codes or months.

You can create a pick list in IBM Planning Analytics Workspace in the following ways:

- By defining the Picklist attribute for a dimension
- By creating a pick list control cube
- By generating a pick list for selected cells

### Important considerations

Cell edits applied through data spreading operations and TurboIntegrator processes are **not** validated. Edits applied through either of these methods can result in cell values that do not conform to valid pick list values. Data spreading can be applied to cells containing pick lists only through the data spreading dialog boxes; data spreading shortcuts cannot be used in cells containing pick lists.

When defining a pick list that contains numeric values, you must use the Cultural Invariant style, which uses a period (.) as a decimal separator. The Cultural Invariant style is equivalent to English style.

## Create a pick list by using the Picklist attribute

Create a pick list either by referencing a set or a dimension in a member attribute that is called `Picklist`, or by adding static values to the `Picklist` attribute.

A pick list that references a set or a dimension contains all the values that are in the set or dimension. If the members change, the values in the pick list also change. A pick list that contains static values contains only those values.

**Note:** A dimension that contains pick lists must be last in the cube. String type members and pick list members must be in the same dimension in a cube.

### Procedure

1. Open the dimension that references the pick list in the dimension editor.
2. Click the **Attributes** tab.
3. Click **Add attribute**.
4. Create a string attribute called `Picklist`.
5. To reference a set, add the following Picklist attribute value:

   ```
   Subset:dimensionname:setname
   ```

   See the following example:

   ```
   Subset:Years:months
   ```

6. To reference a dimension, add the following Picklist attribute value:

   ```
   Dimension:dimensionname
   ```

   For example:

   ```
   Dimension:Employees
   ```

7. To reference the members of a named hierarchy within a dimension, add the following Picklist attribute value:

   ```
   dimension:dimensionname\:hierarchyname
   ```

   See the following example:

   ```
   dimension:Products\:Europe
   ```

8. To create a static pick list, add the following Picklist attribute value:

   ```
   static:value1:value2:value3:value4
   ```

   See the following example:

   ```
   static:none:January:February:March:April:May:June:July:August:September:October:November:Dece
   mber
   ```

   To include a null value at the beginning or in the middle of a static pick list, use two consecutive colons in the pick list definition. For example, the code: `static:value1:value2::value3:value4` results in a pick list with a null value between `value2` and `value3`.

   To include a null value at the end of a static pick list, insert a colon without a following value at the end of the pick list definition. For example, `static:value1:value2::value3:value4:` results in a pick list with a null value that follows `value4`.

### What to do next

To test that the pick list is successfully created, add a view with the dimension that contains the pick list.

## Create a pick list control cube

Use a pick list control cube to define which individual cells a pick list is available from.

### About this task

Using rules, you can define pick lists for any section of a cube, from a single cell to the entire cube.

**Important:** To create pick list control cubes, you must use IBM Planning Analytics version 2.0.8 and later.

A pick list control cube contains the same dimensions as the cube it was created from, with an extra dimension named **}Picklist**. **}Picklist** has one member that is named Value.

### Procedure

1. Right-click the cube that you want to create the pick list control cube for, then click **Edit pick list cube**.

   The rule editor and a view of the new control cube is opened. You can define the pick list either in the view, or in the rule editor.

2. To define the pick list in the view do the following steps:
   a) Change the view of the control cube so that you see the cells that you want to define pick lists for.
   b) Type a pick list definition in the cell.

      You can enter any of the pick list types in the control cube: static, set, or dimension. See “Create a pick list by using the Picklist attribute ” on page 773 for examples.

3. To define the pick list in a rule, do the following steps:
   a) In the rule editor that was created, use a standard rules area definition to specify the cells that you want the pick list to appear in.
   b) Immediately after the area definition, type `=S:`. S is the string qualifier, indicating that the rule applies to string cells.

c) Immediately after the string qualifier, type a pick list definition, which is enclosed in single quotation marks, then enclosed in parentheses. For example, `('static:spring:summer:winter:fall')`

d) Immediately after the pick list definition, type a semi-colon (;) to end the rule statement.

4. To prevent a cell from displaying a pick list, type none in the pick list control cube cell or use `('none')` as the formula in a rules statement. For example, `['season']=S:('none');`.

**Pick list rule examples**

| Rule statement | Description |
|---|---|
| `['size','shirts']=S: ('static:16:17:18');` | This rule statement indicates that any cell identified by the members **size** and **shirts** displays a static pick list with the values 16, 17, and 18. |
| `['size',{'sweaters','vests','jackets'}]= S:('static:XS:S:M:L:XL');` | This rule statement indicates that any cell identified by the member **size** and any of the members **sweaters**, **vests**, or **jackets** displays a static pick list with the values XS, S, M, L, and XL. |
| `['fabric']=S:('dimension:materials');` | This rule statement indicates that any cell that is identified by the member **fabric** displays a pick list with all members in the materials dimension. |

## Generate a pick list for selected cells in the cube viewer

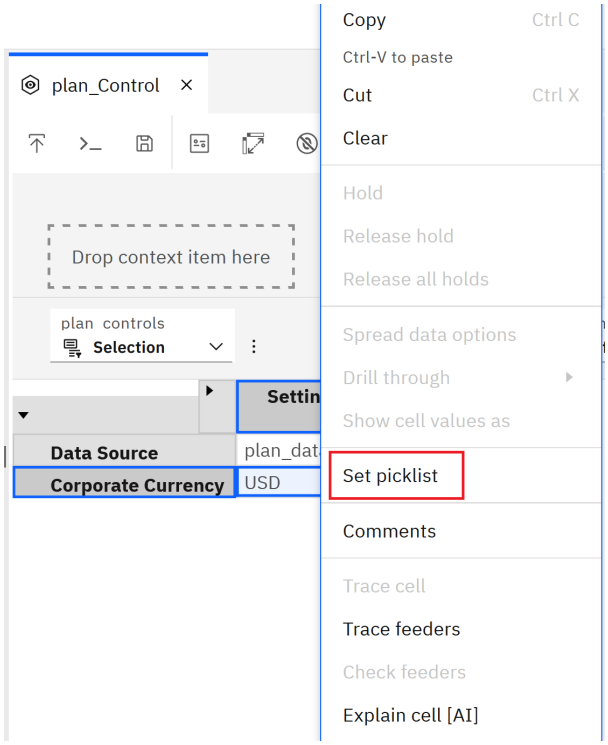
A user in the modeling workbench can create pick lists in the cube viewer by selecting one or more cells and generating a pick list for them.

The type of a pick list that users can generate depends on the type of cell that the pick list is created in. For numeric cells, the pick list is typically a range of numbers that the users can select from. For string cells, the pick list is a static list and the users can quickly enter and edit the list of items. For hierarchies and subsets, the user can select from an existing dimension and hierarchy.

### Procedure

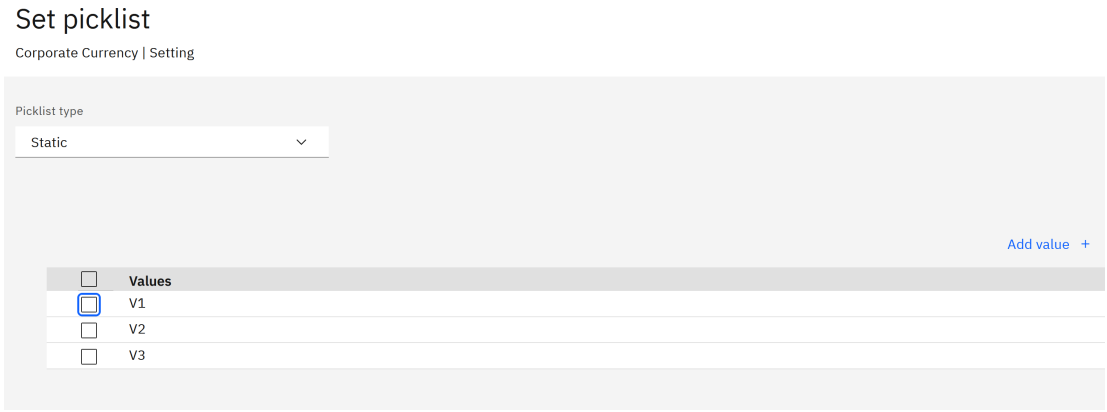
1. In the cube viewer, select a single cell or multiple cells of the same type (string or numeric).
2. Right-click the selection, and in the menu locate the **Set picklist** option.

   If this option is not active, the cell cannot be used to create a pick list. A pick list cannot be created for the following types of cells: rule-based calculations, consolidations, read-only cells, or when the cells are of different types (in a multi-selection).

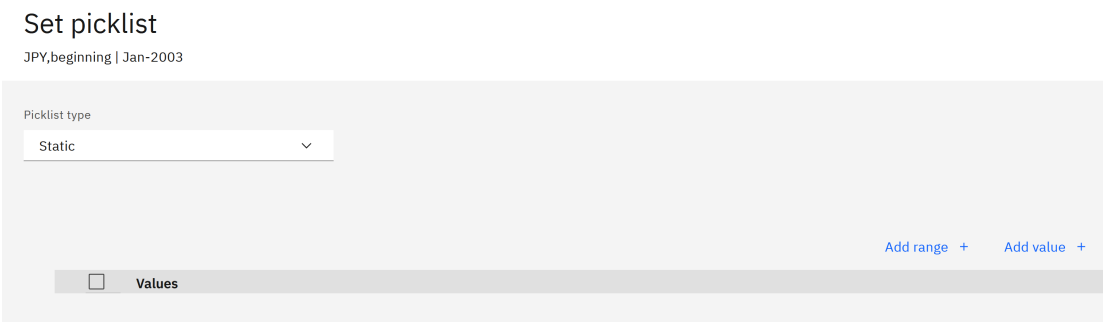



3. Click the **Set picklist** option, and in the dialog box that appears, select the type of a pick list to create:
   - None - clear the pick list if it already exists for the selected cell
   - Static - specify pick list values
   - Hierarchy - use all hierarchy members, or a subset of members, as pick list values
4. If you select the **Static** type, the dialog box that is displayed depends on the cell data type.

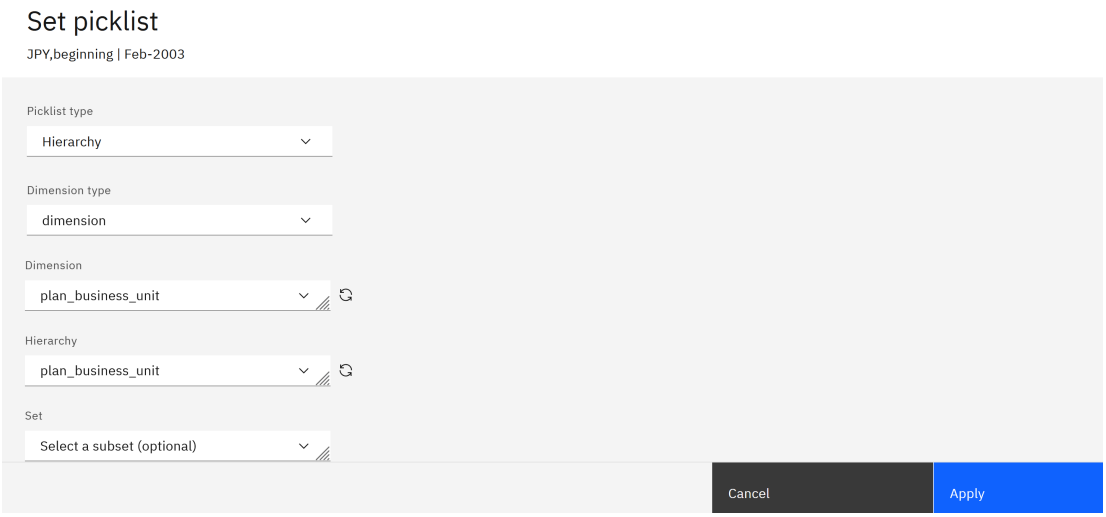
   The following dialog box is displayed for string cells. Click **Add value** to specify the pick list values, and select the value checkboxes.




   The following dialog box is displayed for numeric cells. In this case, you can click **Add value** to type the pick list values, or click **Add range** to specify value ranges.




5. If you select the **Hierarchy** type, the following dialog box is displayed.




Select a dimension first, and then a hierarchy. The dimension type has 2 options, **Dimension** and **Control dimension**. The **Set** option is just a case of the hierarchy option. In this case, the pick list is tied to a subset.

# Dimension editor search and locate in hierarchy

You can determine a member's location using the dimension editor search function.

The dimension editor search function includes a menu item called **Locate**. Searching in the dimension editor acts as a filter to generate a list of members that match the search term. After searching, right-click a member from the generated list and select **Locate** to remove the filters and locate the member in the hierarchy.

**Note:** If the user picks a set (filter), the element is located within the set.

To learn more about the dimension editor, see Dimensions in Planning Analytics Workspace.

# Identify which cubes include a specific dimension

You can now quickly identify which cubes on a Planning Analyticsdatabase include a specific dimension. This is helpful when you are thinking about deleting a dimension, but want to see the impact of the deletion on the cubes on your database.

## About this task

The **View related cubes** option is available only on a modeling workbench. The option is available on the right-click menu anywhere you can see an individual dimension name on the **Databases** tree:

- *DatabaseName* > Dimensions > *DimensionName*
- *DatabaseName* > Cubes > *CubeName* > Dimensions > *DimensionName*

- *DatabaseName* > Control Objects > Control Dimensions > *DimensionName*
- *DatabaseName* > Control Objects > Control Cubes> *CubeName* > Dimensions > *DimensionName*

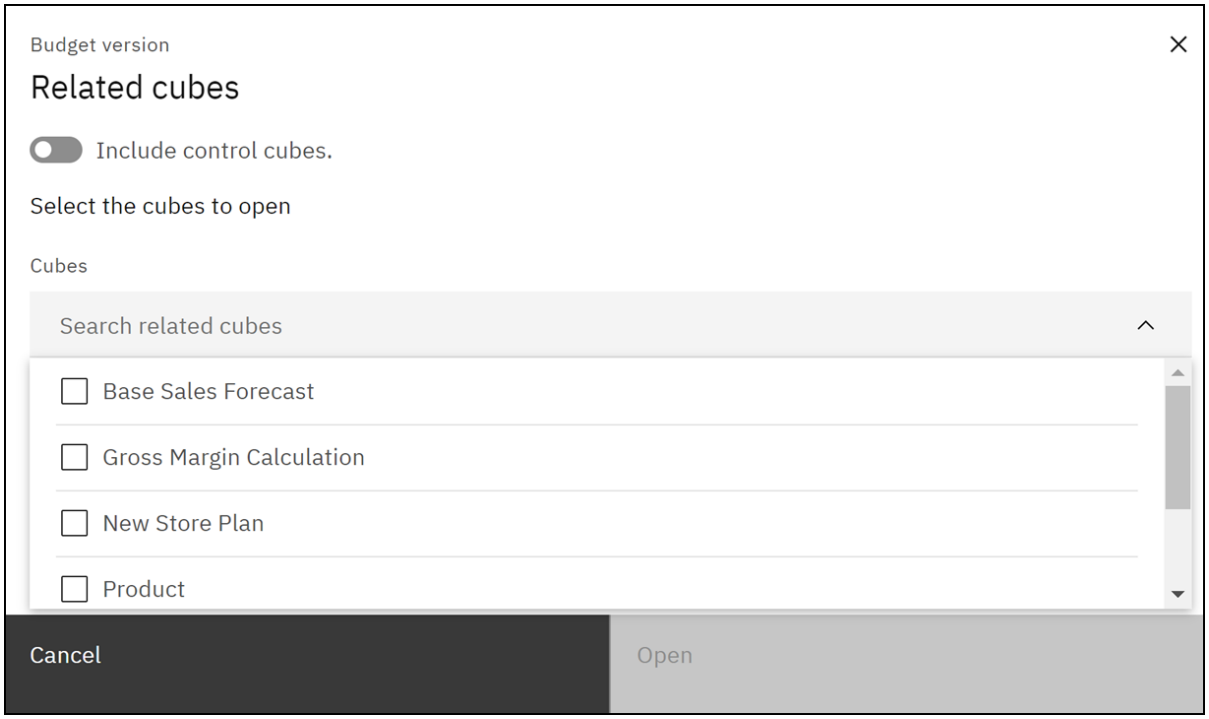
### Procedure

1. Right-click a dimension on the **Databases** tree on a modeling workbench.
2. Click **View related cubes**.

   If the dimension is not included in any cubes, you receive notification and can cancel the procedure.
3. On the **Related cubes** window, toggle the **Include control cubes** option to show or hide control cubes in the list of cubes.
4. In the **Cubes** list, select the cubes you want to open for review.

   If the dimension is included in five or fewer cubes, the cubes are shown in a simple list.

   If the dimension is included in more than five cubes, the cubes are shown in a searchable menu list.



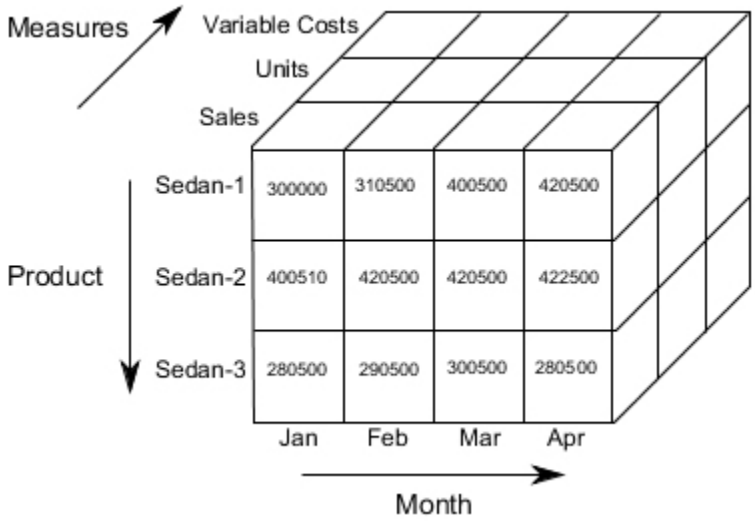

5. Click **Open**.

   Each of the cubes you selected opens for review on a separate tab on the modeling workbench.

# Cubes

IBM Planning Analytics stores the data that you need for planning and analysis in cubes.

Each cube typically has a specific purpose. Suppose that you are analyzing sales; you have a cube that measures the sales for Sedan cars over time. The cube contains three dimensions: Measures, Product, and Month. Each measure, such as Sales, is organized by a product and a month. For example, the cell value 300000 represents the sales of Sedan-1 in the month of January (Jan).




A cube has two or more dimensions. The number of dimensions that there are in a cube depends on the purpose of a cube. For example, a two-dimensional cube is useful as a lookup table; you can store exchange rates in a lookup table.

A revenue planning cube might have the following dimensions: Products, Versions, Regions, Measures, Time.

For a thorough review of cube design best practices, see A Best Practice Guide to Designing TM1 Cubes on the IBM Accelerator Catalog.

## Guidelines for designing cubes

1. List the measures you want to track in your business analysis. Examples of measures include sales amounts, units sold, expenses, acquisition values, and campaign costs.
2. Decide what dimensions you need. Consider the following questions:
   - In most analyses, you track measures over time. What is the base time interval: days, weeks, months?
   - Is there a geographic dimension?
   - Do the measures vary by customer and product?
   - Is there a scenario, or versions dimension. For example, do you want to see the actual figures versus the budget?
3. Determine the hierarchical structure of your dimensions. Every dimension has at least one hierarchy. If multiple hierarchies are enabled, you can use them to see alternative rollups of the data in the same view.
4. Create a list of attributes you want to associate with the dimension members. Examples of attributes include store square footage, customer IDs, and local language versions of member names.
5. Define the display formats for the measures in your cubes. For example, define Gross Margin as a percentage and Sales as a currency amount.
6. The dimension order in a cube is important. The order that you select the dimensions when you create a cube can impact performance. In general, select the dimensions in order of the smallest, sparse dimension first, and the largest, dense dimension last. A dense dimension has a high percentage of values for its members. However, you might find that it is better to put a small dense dimension, such as a versions dimension, before a large, sparse dimension such as a products dimension.
7. If the dimension contains members that have the text type, or is a pick list, this dimension must be last in the cube. Text type members and pick list members must be in the same dimension.

# Create a cube

You can create cubes when you are a Modeler or an Administrator. Cubes are created and stored on a specific database.

## Create a cube in Planning Analytics Workspace

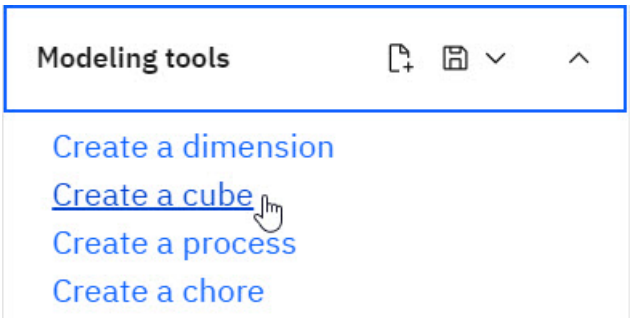
### About this task

As of Planning Analytics Workspace 2.0.75, cubes can be created only on a Modeling workbench. You cannot create a cube in a book.

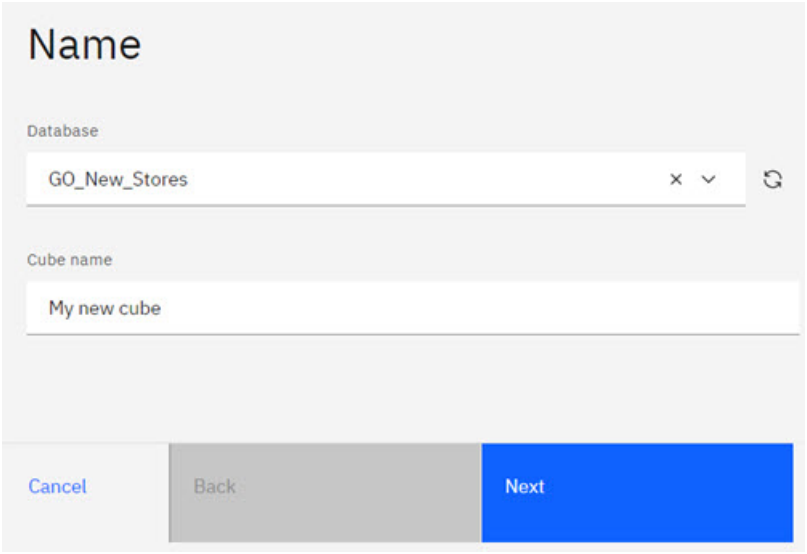
**Note:** When you create a cube using the REST API, transaction logging is enabled on the new cube. The Planning Analytics Workspace modeling workbench uses the REST API for all interactions with TM1. However, if you create a cube with the CubeCreate TurboIntegrator function, transaction logging is not enabled. To enable transaction logging, use the CubeSetLogChanges TurboIntegrator function.

### Procedure

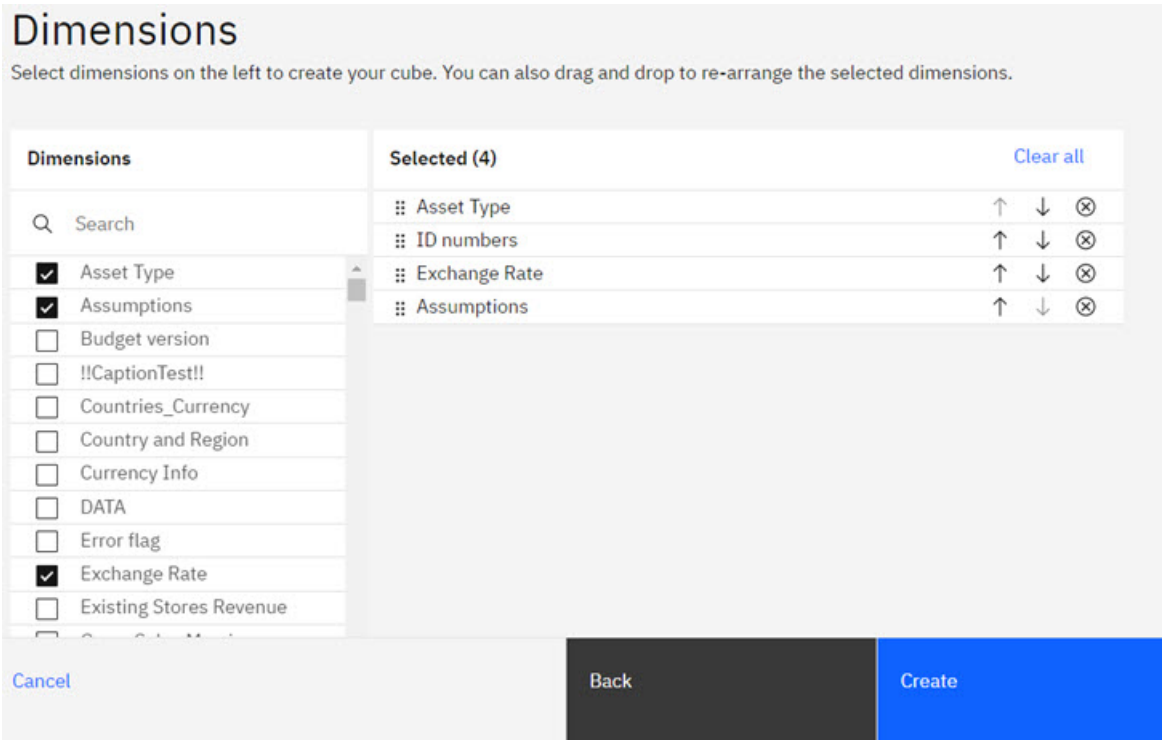
1. Open a Modeling workbench.
2. Expand the **Modeling tools** menu, then click **Create a cube**.




3. Select the database on which you want to create the cube.
4. Enter a name for the cube.




5. Click **Next**.
6. Select the dimensions you want to include in the cube from the **Dimensions** list. When you select a dimension, it is added to the **Selected** list.




7. Optionally, you can change the order of dimensions in the cube by clicking a dimension handle and dragging it to a new position or by clicking the **Up** or **Down** arrow next to a dimension.
8. Click **Create** to create the new cube.

### Create a new cube based on the structure of an existing cube

If a cube exists with the structure and dimensionality that you want to use in a new cube, the **Create from cube** option provides a handy shortcut for cube creation.

#### Procedure

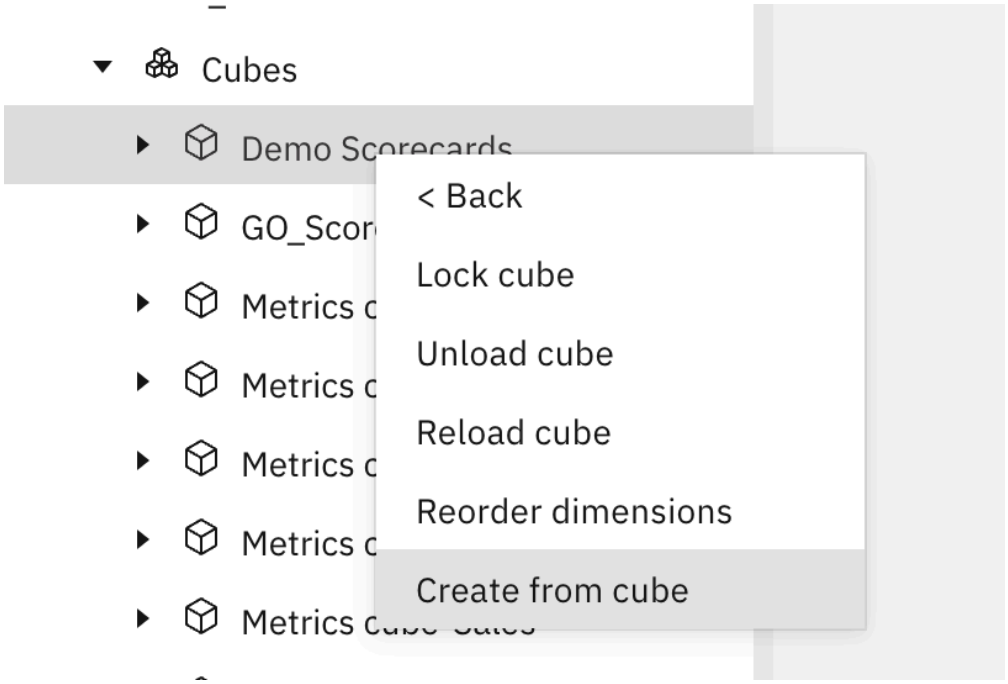
1. Open a Modeling workbench.
2. In the **Databases** tree, locate the existing cube that you want to use as the basis for a new cube.
3. Right-click the existing cube, then click **Create from cube**.
4. Enter a new **Cube name**, then click **Next**. The dimensions from the existing cube are preselected and ordered as specified in the existing cube.
5. Optionally, you can select additional dimension to include in the new cube, remove preselected dimension, or change the order of dimensions for the new cube.
6. Click **Create**.

## Recreate a cube from an existing cube

You can replace an existing cube with a new cube from the **Create cube** menu.

#### Procedure

1. In the **Databases** tree, right-click on the cube that you want to replace.
2. Click **Create from cube**.




3. In the **Create cube** menu, you can change the database and name of the cube. Then, click the check box next to **Replace cube**.
4. Click **Next**.
5. In the **Dimensions** menu, you can add or remove dimensions from the cube you are creating.
6. Click **Create**. A warning will appear, explaining that deleted data is not recoverable.
7. Click **OK**. The existing cube is deleted, and a new one is created with the same name.

## Export data from a cube or view

You can export data from a cube or view to a `.csv` file. The exported file is saved to your local `Downloads` directory.

### About this task

You can initiate a data export from several places on the modeling workbench **Databases** tree:

**Database**
Right-click a database, then click **Import/Export** > **Export data**.

**Cubes node for a database**
Right-click the **Cubes** node for a database, then click **Export data**.

**Cube**
Right-click a specific cube, then click **Import/Export** > **Export data**.

**View**
Right-click a specific view, then click **Export data**.

### Procedure

1. Start the data export from one of the available locations on the modeling workbench **Databases** tree.
2. If necessary, select the **Cube** from which you want to export data. This is required only if you initiate an export from a database or from the **Cubes** node on a database.
3. By default, the name for the exported `.csv` file is the same as the source cube. If you want to use a different name, enter it in the **File name** box. You do not need to include a file extension.
4. Choose whether you want to export all cube/view data or want to refine the scope of the data that is exported.

- Click **Export** to export all cube or view data, *exclusive of consolidated values, rules-derived values, and zero/blank values*.

  If you initiated data export from a database, **Cubes** node, or individual cube, all data from the source cube is exported.

  If you initiated data export from an individual view, only the data included in the source view is exported.
- Click **Next** to set options that define the data that is exported.

5. Optionally, select a **View** as the basis for the export.

   When you select a view, the **Dimension elements** grid updates to reflect the view configuration.
6. Use any combination of the **Hierarchy**, **Set**, and **Member** menus to further define the scope of data to export for each dimension.

   For example, you can:

   - specify only a hierarchy to include all members of the hierarchy in the data export.
   - specify a hierarchy and set to include all members of the set in the data export.
   - specify a hierarchy, set, and single member to include only the selected member in the data export.
7. If you want to filter the numeric values that are included in the export, click the **Numeric values operator** menu and select an expression. Enter the values that are required to evaluate the expression.
8. If you want to filter the string values that are included in the export, click the **String values operator** menu and select an expression. Enter the values that are required to evaluate the expression.
9. By default, an export does not include consolidated values, rules-derived values, or zero/blank values. If you want to include any of these types of values in an export, clear the check box next to the appropriate **Skip** option.
10. Click **Export**.

    The exported data is saved in a `.csv` file in your browser's **Downloads** directory, using the name specified in step 3.

# Importing data into a cube

Modelers can import data into a new or existing cube.

You can create a process and a chore so that you can schedule regular updates to the data in the cube, and you can import data into control cubes.

## Before you begin

You must be a member of the ADMIN group for the TM1 database to be able to import data into cubes. See “Add users to user groups” on page 850.

Your import data must be correctly structured and formatted to get the correct results, see “Cube import file format” on page 786.

## Procedure

1. There are multiple ways to begin importing data on a modeling workbench or in a book. When importing data in a book, you must be in edit mode.
   - Right-click a database on the **Databases** tree, then click **Import/Export** > **Import data**.
   - Right-click a **Cubes** node on the **Databases** tree, then click **Import data**.
   - Right-click a cube on the **Databases** tree (either a regular data cube or a control cube), then click **Import/Export** > **Import data**.
2. Specify a **Cube** action.
   - To import data into an existing cube, select the cube from the list of available cubes.

- To import data into a new cube, select the **New cube** option and enter a name for the new cube.

3.
4. Specify a **Data source**.
   - For a delimited text file source, select **File** from the menu.

     To import a file that is external to your Planning Analytics database, enable the **Upload file** option, then drag the file onto the drop zone or click the link to navigate to the file.

     **Tip:** In IBM Planning Analytics Workspace Local, drag and drop requires the Planning Analytics Administration agent to be configured and running.

     To import data from a file that is stored in File Manager on your Planning Analytics database, disable the **Upload file** option, then click **Select file**.
   - For an ODBC source, select **Database connection**. Select the connection you want to use from the **Database connection** list, then provide authentication credentials if required. If the database supports Unicode, enable the **Unicode** option.

     TM1 Database 12 requires the ODBC Data Connector to use a database connection for a data source.
5. Click **Next**.
6. Preview the data source.
   - For a file source, preview the file and confirm that the correct **Delimiter**, **Quote character**, and **Header records** formatting options are applied. If necessary, modify any of the formatting options and then click **Refresh** to see the impact on the preview.
   - For an ODBC source, enter a query to extract data, then click **Refresh** to preview the results.
7. Click **Next**.

   The remainder of the import procedure varies depending on whether you are importing data into an existing cube or importing into a new cube.

## Importing data into an existing cube

When you import data into an existing cube, you must specify how to handle existing cube data and map columns in your source data file to existing cube dimensions.

### Procedure

1. Select a **Before loading data** option to specify how to handle existing cube data.
   - Select **Do not clear** to leave existing cube data as-is.
   - Select **Clear all data** to reset all values in the cube to 0.
   - Select **Clear data from view** and then select a view. All values in the selected view are set to 0.
2. Select a **While loading data** option to specify how to handle values while loading data from your source file.
   - Select **Overwrite existing data** to overwrite existing values in the cube. If there is more than one value in the import file for a cell, the last value is written to the cube. This option has no impact on string cells. If the cell contains text, the current value is always overwritten.
   - Select **Accumulate with existing data** to add the values that are imported to existing cell values. If there is more than one value in the import file for a cell, all the values are accumulated. This option has no impact on string cells. If the cell contains text, the current value is always overwritten.
3. If necessary, update the **Decimal separator** and **Thousands separator**.
4. You can optionally save the import configuration as a process and create a chore that you can schedule for automatic process execution.
   a) Select the **Save process as** option and enter a name for the new process.

      The process is created on your database and is available in the **Processes** node of the Data tree.

b) Select the **Schedule process as** option and enter a name for the new chore.

The chore is created on your database and is available in the **Chores** node of the Data tree, but the chore frequency is not set and the chore is not enabled. See Create and edit chores for more information on scheduling chores.

5. Map the dimensions in the cube to columns in the data source.

For each cube dimension, click **Select mapping** and pick the column in the source that corresponds to the dimension.

6. Specify **Data** mapping.
   a) Specify the type of data contained in your source file, **Numeric** or **String**.
   b) Click **Select mapping** and pick the column in the source file that contains data values.
7. Click **Import**.

## Importing data into a new cube

When you import data into an new cube you must specify a dimension mapping format, specify how to handle data values, and map columns in your source data file to new or existing cube dimensions.

### Procedure

1. Select the **Dimension mapping format** that matches the structure of your source file.
   - If your source contains only leaf members for each dimension, select **Leaf only**.
   - If your source contains nothing more complex than a simple parent-child structure for dimensions, select **Parent-Child**. For example, you might have two columns in your source, Regions and Cities, where Regions is the parent, and Cities is the child.
   - If you are importing any dimensions with more than two levels, select **Multi-level**. An example of a multi-level dimension is one with the following columns: Product, Product type, Product category, Total product.

   **Tip:** Your selection should reflect the most complex dimension structure that is present in your source file. For example, if your source contains multiple levels of consolidation for some dimensions, but only leaf members for other dimensions, you should select **Multi-level**.
2. Select a **While loading data** option to specify how to handle values while loading data from your source file.
   - Select **Overwrite existing data** to overwrite existing values in the cube. If there is more than one value in the import file for a cell, the last value is written to the cube. This option has no impact on string cells. If the cell contains text, the current value is always overwritten.
   - Select **Accumulate with existing data** to add the values that are imported to existing cell values. If there is more than one value in the import file for a cell, all the values are accumulated. This option has no impact on string cells. If the cell contains text, the current value is always overwritten.
3. If necessary, update the **Decimal separator** and **Thousands separator**.
4. You can optionally save the import configuration as a process and create a chore that you can schedule for automatic process execution.
   a) Select the **Save process as** option and enter a name for the new process.

   The process is created on your database and is available in the **Processes** node of the Data tree.
   b) Select the **Schedule process as** option and enter a name for the new chore.

   The chore is created on your database and is available in the **Chores** node of the Data tree, but the chore frequency is not set and the chore is not enabled. See Create and edit chores for more information on scheduling chores.

   Now you can map columns in your data source to new or existing dimensions on your database and identify the column(s) that contain data values.

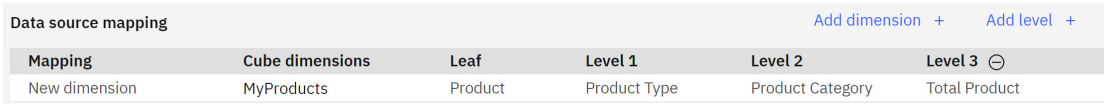
5. The default **Data source mapping** panel is configured for a data source with two dimensions. If your data source contains more than two dimensions, click **Add dimension** until the **Data source mapping** panel is configured to accommodate all dimensions in your source.
6. For each dimension represented in you source, use the **Mapping** column to specify if the dimension maps to a **New Dimension**, exiting **Dimension**, or existing **Control Dimension**.
7. If mapping to a new dimension, enter a dimension name in the **Cube dimensions** column. If mapping to an existing dimension, select a dimension name in the **Cube dimensions** column.
8. Define mappings for all of the dimensions represented in your data source.
   - For **Leaf only** dimension mapping, in the **Child** column, select the column in your source that contains the leaf members.
   - For **Parent-Child** dimension mapping, select the column of data that contains the parent members in the **Parent** column. Then, in the **Child** column, select the column of data that contains the child members.
   - For **Multi-level** dimension mapping, complete these steps.

   a) When you specify **Multi-level** dimension mapping, the default **Data source mapping** panel is configured for a data source with two levels of consolidation, **Level 1** and **Level 2**. If your data source contains more than two levels of consolidation, click **Add level** until the **Data source mapping** panel is configured to reflect the levels of consolidation that are present in your source.

   b) In the **Leaf** column, select the column of data that contains the leaf members for the dimension.

   c) In each of the **Level** columns, select the column in your data source that corresponds to the level of consolidation.

   For example, if your source contains the **`Product`** column at the leaf level, **`Product Type`** at the first level of consolidation, **`Product Category`** at the second level of consolidation, and **`Product Total`** at the third level of consolidation, the configuration would look like this.




9. Specify **Data** mapping.
   a) Specify the type of data contained in your source file, **Numeric** or **String**.
   b) Click **Select mapping** and pick the column in the source file that contains data values.
10. Click **Import**.

    **Tip:** If the **Import** button is disabled, it means you haven't mapped all the columns in your source. Click any of the current mappings to see a list of the columns in your data source. Any column without a corresponding check mark has not been mapped.

# Cube import file format

When you import a file into a cube, the file must be structured and formatted in a certain way to get the results that you want.

## File formats

The import data must be saved in a text file format, such as comma-separated (CSV) or with any other standard delimiter such as tab, space, semicolon, colon, vertical line, or caret (^).

**Note:** Files that contain a Byte Order Marker (BOM) might be corrupted when the file is imported.

## Structure of the data

Each column in the text file represents either a dimension or the values.

**Value data**

Value data is the data that goes into cells. Values can either be numbers or string values. You can import only one column of value data.

**Dimensions**

The items in the dimension columns must match up with the dimension member names in the cube, otherwise the row is not imported, and an error such as

```
line (300): Invalid key: Dimension Name: "Products", Element Name (Key): "Mountain Man
Analog"
```

is displayed.

If you are creating a new cube, the file must contain one dimension per column.

The column headings do not need to match the dimension names.

The column order in the text file does not need to match the order of dimensions in the cube, and you select which columns of data to import. The column headings do not need to match the dimension names.

If several rows in the text file target the same cell, the value in the last row is imported, and all other values are ignored unless the **Accumulate with existing data** option is selected. **Accumulate with existing data** sums up values as they are imported, adding the values that are being imported to the existing contents of the cell.

### Importing data into a cube

Say that you have a cube with three dimensions: Cities, Products, Time_months. You want to import a .csv text file with five columns that contains sales data. The first eight lines of the text file, including the header row, are shown.

```
Product,Month,Branch,Qty,Sales
Capri Orange Women's,Nov-15,Boston,3976,3976
Inferno Blue Men's,Jul-15,Bury St Edmunds,2719,122355
TX Green Men's,Aug-15,Leiston,3078,49248
Canyon Mule Climber Backpack Red 32000 cc,Apr-15,Ipswich,4801,388881
Capri Orange Women's,Sep-15,Bridgeport,46,46
Glacier Deluxe Black Unspecified,Sep-15,London,3839,122848
Cat Eye Orange Women's,Jun-15,Dundee,3221,235133
```

1. The header row contains five columns: Product, Month, Branch, Qty, and Sales
2. Map the following columns to the dimension names as listed:
   - Map the Product column to the Products dimension
   - Map the Month column to the Time_months dimension
   - Map the Branch column to the Cities dimension
3. Decide which values you want to bring in, either Qty, or Sales. In this case you want to import the Sales data, so select **Numeric Values** for Sales. The Qty column of data is ignored.
4. You can now import the file.

### Creating a cube with multi-level dimensions

The following example CSV file has 16 columns of data. Some of the dimensions have 3 or more levels.

```
Organization, Region, Total Company, Channel, Total Channel, Product, Product Type, Product
Category, Total Product, Year, Month, Quarter, Total Year, Version, Sales Measure, Value
Massachusetts, East Region, Total Company, Retail, Total Channel, 3G 32Gb, 3G Smart Phones,
Phones, Total Product, 2018, Jan, Q1, Total Year, Budget, Unit Net Sales Price, 70
Florida, East Region, Total Company, Distribution, Total Channel, 3G 16Gb, 3G Smart Phones,
Phones, Total Product, 2018, Sep, Q3, Total Year, Budget, Volume - Units, 5,58
Michigan, Central Region, Total Company, Retail, Total Channel, SP 2110, Desktops, PCs, Total
Product, 2018, Dec, Q4, Total Year, Actual, Direct COGS, 15592
Maryland, East Region, Total Company, Internet, Total Channel, 8 64 Gb, 8 Inch Tablets,
Tablets, Total Product, 2018, Feb, Q1, Total Year, Budget, Volume - Units, 26,97
Illinois, Central Region, Total Company, Distribution, Total Channel, 3G 32Gb, 3G Smart Phones,
Phones, Total Product, 2019, Feb, Q1, Total Year, Actual, Unit Direct Cost, 47,141,464
California, West Region, Total Company, Retail, Total Channel, SP 2110, Desktops, PCs, Total
```

```
Product, 2019, Nov, Q4, Total Year, Budget, Indirect COGS, 100
California, West Region, Total Company, Retail, Total Channel, SP 2110, Desktops, PCs, Total
Product, 2019, Nov, Q4, Total Year, Actual, Unit Net Sales Price, 341454
```

To create a new cube and dimensions with this file, do the following steps:

1. Select the **Multi-level import** option.
2. Click + to add a level.
3. Click **Add dimension** four times.
4. Map the data as shown in the following table. You might need to change the names in the cube dimensions name if dimensions exist with that name.

| Cube dimensions | Leaf | Level 1 | Level 2 | Level 3 |
|---|---|---|---|---|
| Organization | Organization | Region | Total Company | |
| Channel | Channel | Total Channel | | |
| Product | Product | Product Type | Product Category | Total Product |
| Time | Month | Quarter | Year | Total Year |
| Version | Version | | | |
| Sales | Sales Measure | | | |
| Data | Numeric | Value | | |

5. Click **Create cube**.

# Optimize the order of dimensions in a cube

You can change the order of dimensions in a cube in IBM Planning Analytics Workspace and view the percent change in memory. Changing the order of dimensions can reduce memory consumption and improve performance.

When you optimize the order of dimensions in a cube, Planning Analytics does not change the actual order of dimensions in the cube structure. Instead, it changes the way dimensions are ordered internally on the database. Because the cube structure is not changed, any rules, functions, or applications that reference the cube remain valid.

As you change the order of dimensions, you can instantly view a report detailing the impact that your changes have on cube memory consumption.

For the following reasons, experiment with the order of dimensions in a cube only in a development environment:

- Significant memory resources are required for the database to reconfigure the order of dimensions in a cube. During the reordering process, the temporary RAM on the database increases by a factor of two for the cube that you are reordering. For example, a 50 MB cube requires 100 MB of RAM to reconfigure.
- Reordering puts a read lock on the database, locking all user requests while the reorder is performed.

When you have a satisfactory dimension order in your development environment, modify the order of dimensions in your production environment. Remember that there will be an impact on memory consumption and server availability while the reorder is being executed.

## About this task

You can change the order of dimensions for cubes on Planning Analytics databases version 2.0.6 or later. If a cube is on a database before version 2.0.6, the **Reorder dimensions** option is unavailable when you right-click the cube in the Databases tree.

You must be a Planning Analytics administrator or modeler to optimize the order of dimensions in cubes. When you optimize the order of dimensions in a cube, you cannot move a dimension that includes string

members from the last position. If a dimension with string members exists in your cube, it must be the last dimension in the ordering.

You can reorder cube dimension from a modeling workbench.

You cannot reorder dimensions in control cubes.

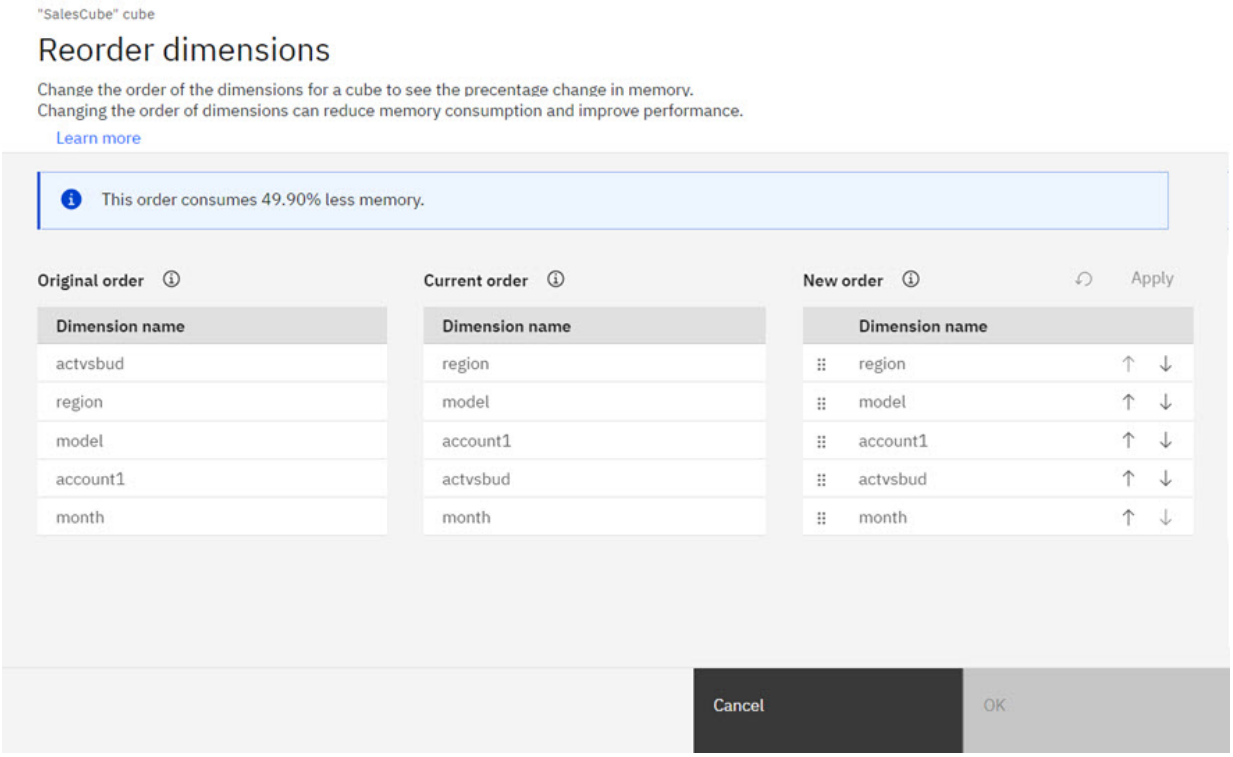
## Procedure

1. In the **Databases** tree of a modeling workbench, right-click the cube that you want to optimize.
2. Click **Reorder dimensions**.

   The **Reorder dimensions** page opens. This screen shows the original order of dimensions, the current ordering of dimensions on the database, and a list of cube dimensions that you can manipulate to set a new order.
3. Select a dimension in the **New order** list.
4. Click the up ↑ or down ↓ arrow to change the order of the dimension in the cube.
5. Click **Apply**.

   When you click **Apply**, the **New order** is applied to the cube and the **Current order** list updates. Note that after you click **Apply**, the **Apply** and **OK** options are disabled. This is because clicking **Apply** results in the **Current order** being set to the specified **New order**; there is no difference between current and new so there is no action to apply

   Review the message that indicates the change in memory consumption to determine if the new order is more efficient.




6. Repeat steps 3 - 5 until you achieve the most efficient ordering of dimensions.

### Optimize the order of dimensions without reviewing the change in memory consumption

If you already know the order of dimensions that you want to use and don't need to review the impact of different dimension orderings, you can set the order in the **New order** list and then click **OK**.

#### About this task

If you use this option, you don't see a message indicating the change in memory consumption on the **Reorder dimensions** page. Instead, notifications on the modeling workbench inform you when the reordering is applied and show the change in memory consumption.

## Lock and unlock a cube

Lock a cube to prevent updates to the data in the cube.

Analysts with lock permissions can lock a cube. The user who locked the cube or any user in a group with ADMIN privilege can also unlock it.

#### Procedure

1. Right-click the cube in the Data tree and select **Lock cube**.

   A locked cube has the lock icon. The cells in a locked cube view are gray when the view is refreshed.
2. To unlock the cube, right-click the cube and select **Unlock cube**.

## Unload and reload a cube from memory

You can unload a cube from memory to temporarily reduce RAM consumption or to assist in the development and troubleshooting of rules feeders. Unloading a cube also unloads any views of the cube from memory.

#### About this task

While a cube is unloaded, any reference to or request for data in the cube will cause the cube to be automatically reloaded, maintaining data availability.

You must be an administrator or modeler to unload or reload cubes.

#### Procedure

1. Make sure that you are in edit mode.
2. In the Data tree, right-click the cube you want to unload, then click **Unload cube**.
3. Click **OK** when prompted for confirmation.

   To manually reload a cube into memory, right-click the cube, then click **Reload cube**.

## Delete a cube or a view

You can delete cubes or views from the data tree when you are a Modeler or an Administrator.

#### About this task

You cannot delete control cubes, which are used to run and maintain your IBM Planning Analytics system. When you delete a cube, you also delete all data and any rules or views that are associated with the cube.

#### Procedure

1. In the Data tree, navigate to the database that contains the cube or view that you want to delete.
2. Expand the database to reveal the dimensions, cubes, views, and other groups.

3. To delete a cube, click **Cubes**, right-click the cube that you want to delete, then click **Delete cube**.
4. To delete a view from the database, click **Views**, right-click the view that you want to delete, then click **Delete view**.

## Manage cube properties

Cube properties provide information about cubes that are used in different processes. Some of these properties cannot be edited directly, they are updated by the system.

To view and manage cube properties, click  on the cube view toolbar.

**Note:**

Cube properties are stored in the `}CubeProperties` control cube. In some cases, this control cube may not be present on a Planning Analytics database. To create one, an administrator can run the following code in the Metadata section of a TurboIntegrator process:

```
CubeCreate( '}CubeProperties', '}Cubes','}CubeProperties' );
```

| Element | Description |
|---|---|
| REPSTATUS | If a cube is replicated on the database, the value of RepStatus is `Copied`, otherwise this property value is empty.<br>For more information, see Replicating Cubes(https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_op.2.0.0.doc/c_replicatingcubes_n110007.html).. |
| SYNCVIEWS | For replicated cubes, this property indicates whether views are synchronized when the associated cube is synchronized. |
| SYNCRULE | For replicated cubes, this property indicates whether rules are synchronized when the associated cube is synchronized. |
| DEMANDLOAD | Indicates that a cube is automatically loaded when the database starts or is loaded 'on demand' only when a cube value is requested. By default, all cubes are loaded into memory when a database starts. While this allows fast access to data, it can consume significant resources. If your database contains infrequently accessed cubes, you can conserve resources by setting those cubes to load only when a client attempts to access the cube data.<br>When a cube is loaded on demand, the value of the DemandLoad property is YES, otherwise the property value is NO. |
| TIME_DIMENSION | If a time dimension is defined for a cube, this property stores the name of the dimension. |
| MEASURES_DIMENSION | If a measures dimension has been defined for a given cube, this property stores the name of the dimension. |
| LOCK | When a cube is locked, this property stores the name of the client that has locked the cube. |

| Element | Description |
|---|---|
| VMT | For each cube, this property defines the time threshold, in seconds, beyond which the algorithm that stores Stargate views is triggered.<br>If the time required to calculate a cube view surpasses the specified threshold, IBM Planning Analytics attempts to store a Stargate view. If not enough memory is available to store the Stargate view, Planning Analytics purges the oldest Stargate view that is not currently in use, and continues to purge views in this manner until sufficient memory is made available.<br>If no VMM value is specified the default value is 5 seconds. The valid range is 1 - 259,200 seconds. |
| VMM | For each cube, this property determines the amount of RAM reserved on the server for the storage of Stargate views. The more memory made available for Stargate views, the better the performance is. You must, however, make sure sufficient memory is available for the TM1 server to load all cubes.<br>The value of VMM is expressed in kilobytes. If no VMM value is specified the default value is 128 kilobytes.<br>The valid range for VMM is 0 - 2,147,483,647 KB. The actual upper limit of VMM is determined by the amount of RAM available on your system. |
| REPSCRNAME | For replicated cubes, this property stores the name of the source cube. |
| REPLICATION | For replicated cubes, this property stores the name of the replication connection with which the replicated cube is associated. |
| LOGGING | Indicates that logging is enabled for a cube. When cube logging is enabled, the value of this property is YES, otherwise the value is NO. |
| VIRTUALCUBE | Stores the name of the virtual cube. A virtual cube has no data loaded in it, but references data in other cubes by using rules. |
| SAPCLIENTID | Used when connecting to SAP. The client number that corresponds to the UI version on the SAP server to which you are connected. |
| SAPCLIENTLANG | Used when connecting to SAP. The language setting for the SAP system. |
| PROVIDERSTRING | |
| SAPVARIABLESCLAUSE | |
| SLICERMEMBERS | |

| Element | Description |
|---|---|
| DATARESERVATIONMODE | Data Reservation (DR) is a server-related feature in TM1 that allows you to configure exclusive write-access to regions of a cube for individual users. By default, Data Reservation is not enabled. An administrator must enable and configure the feature before you can use the related TurboIntegrator and API functions to manage Data Reservations.<br><br>You can enter the following values in English only. These keyword values are not translated:<br><br>**OFF**<br>Turns off the Data Reservation feature for the specific cube. The default value is OFF.<br><br>**REQUIRED**<br>This mode disables write access for all users for the entire cube and requires you to explicitly assign Data Reservations for any user that needs to write to this cube. For example, a user must have a Data Reservation on a cell if they want to write to that cell.<br><br>**REQUIREDSHARED**<br>This mode is a variation of the REQUIRED mode that allows Data Reservations for different users to overlap. All other aspects of this mode behave the same as REQUIRED mode.<br><br>REQUIREDSHARED mode was implemented to accommodate overlapping requests that use multi-node edit capability in IBM Cognos TM1 Applications. This mode is the default assigned DR mode on all cubes represented by Cube Views or Manual Dependencies in TM1 Applications.<br><br>In REQUIRED mode, the TM1 database restricts write access to a slice by allowing only a single user to have a reservation for a node at any one time. In REQUIREDSHARED mode the application must enforce this restriction if necessary.<br><br>**ALLOWED**<br>Maintains write access, based on security, for all users across the entire cube, but allows you to selectively restrict write access to an area of the cube by assigning Data Reservations to individual users.<br><br>For example, ALLOWED mode sets aside a section of a cube for a specific user while keeping write access available for all other users to the rest of the cube. |
| ALLOWPERSISTENTHOLDS | Specifies whether cells continue to be excluded from data spreading after the user logs out and logs in again. |
| CALCULATIONTHRESHOLD | Sets the threshold for the calculation cache in terms of number of calculation steps. Setting the cube property overrides the global value CalculationThresholdForStorage that is defined in the `tm1s.cfg` file. |
| RULE_STATS | This property determines whether performance statistics are collected for the rules that are associated with the cube.<br><br>To enable statistic collection, set the Rule_Stats property to YES. To disable statistic collection, set the property to NO, the default value).<br><br>Rule_Stats is a dynamic property, meaning that it does not require a server restart to take effect. However, it can take up to 60 seconds for a dynamic property change to be applied on the TM1 server. |

| Element | Description |
|---|---|
| MAINTAIN_DETAILED_ FEEDER_MEMORY_STATS | A value of 'YES' for the ( cubeName, propertyName ) tuple turns on the detailed feeder memory used calculation.<br>**Note:** The detailed feeder memory used calculation is intended to be used during pre-production as an aid to help design the cube. When this step is complete, the property value should be switched to 'NO', so that the much faster feeder memory used calculation is used during production. |

## Manage cube attributes

Cube attributes provide information about the purpose of cubes. Attributes can also be used to identify a cube type or to enable cube names to be displayed in different languages.

### About this task

You can view and edit the values for dimension attributes, but you cannot create or delete attributes in the editor. Attributes for cubes are created in a TurboIntegrator process that uses the cube attribute functions.

### Procedure

1. On the **Database** tree on a modeling workbench, expand a database and right-click the **Cubes** node.
2. Click **Cube Attributes**.

   A modified view of the }CubeAttributes control cube opens, with individual cubes on the row axis and attributes on the column axis.
3. Click a cell at the intersection of a cube and an attribute, then enter or modify the attribute value.

   You can use cut, copy, and paste to manage attribute values. You can also use the standard view options to manage sets and to select multiple cells, rows, or columns in the editor.

# Rules

You can create complex data calculations in IBM Planning Analytics with business rules.

The most common calculations in Planning Analytics involve aggregating data along a dimension. These calculations are performed automatically based on your dimension hierarchy definitions. However, in many models you need to perform calculations that do not involve aggregating, such as cost allocations and exchange translations. With business rules, you can create formulas to perform these calculations.

With business rules, you can perform the following tasks:

- Multiply prices by units to yield the sales amounts.
- Override consolidations when necessary. For example, you can prevent a quarterly price from displaying a tally of individual monthly prices.
- Use data in one cube to perform calculations in another cube, or share data between cubes. For example, you can pull sales data into a cube that contains Profit and Loss information.
- Assign the same values to multiple cells.

For more information about business rules, see the *TM1 Rules* documentation.

# Create and edit rules

You can create and edit business rules when you are a Modeler or an Administrator.

## Procedure

1. Open Planning Analytics Workspace and log in with a user that has the Modeler or Administrator role.
2. On a modeling workbench, navigate the **Databases** tree and expand the database where you want to create or edit a rule.
3. Expand the **Cubes** node to reveal all cubes on the database.
4. Right-click the cube for which you want to create or edit a rule.
   a) Click **Create business rules** to create a new rule.
   b) Click **Edit business rules** to edit an existing rule.
5. Create or edit your rules in the rules editor.
6. Click the **Save** icon.

# Use the rules editor in Planning Analytics Workspace

These features help you create and manage rules in the Planning Analytics Workspace rules editor.

## Open the rules editor in a separate window

To open an the editor in a separate window, click **Open tab in floating window** ⧉.

For more information, see "Open editors in a separate window" on page 730.

## Auto-completion

You can use the CTRL+SPACE keystroke combination to choose from a list of applicable items in the rules editor. If you use CTRL+SPACE after typing one or more characters of an object name, the list presents all items that match the characters you typed.

When you use the auto-completion keystroke combination within an area definition (inside [ ] brackets) or formula component of a rule statement, you are presented with valid selections.

If the space within the brackets is empty, auto-completion presents a list of all dimensions and members that are part of the cube that is associated with the rule. Dimensions are listed first in alphabetical order, followed by all members from all dimensions in alphabetical order. Members are not grouped according to the dimensions to which they belong.

A fully qualified member reference in an area definition or formula component contains three parts: a dimension specification, a hierarchy specification, and the member name.

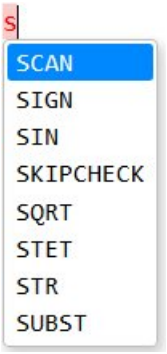
```
DimName:'HierarchyName':'MemberName'
```

If hierarchy support is not enabled in your database, the hierarchy specification is not necessary. All three parts are not necessary to create a member reference; it is usually sufficient to use the member name when the member name is unique across all dimension. However, if you are creating a fully qualified reference, auto-completion presents you with just the items that are relevant to the current partial reference.

For example, if you enter a dimension name in the area definition, then use the CTRL+SPACE keystroke combination, you see a list of the hierarchies and members that are part of the dimension.

Similarly, if your area definition includes a dimension name and a hierarchy name, you see a list of the members that are part of the hierarchy when you use the auto-completion feature.

If you begin entering a function name in the rules editor outside of an area definition or formula component, and then use the CTRL+SPACE keystroke combination, you can pick from a list of rules functions that match the characters you typed.




## Function list

You can click on the rules editor to quickly insert a rules function into your business rules.

When you click , you see a list of function categories. Click a function category to see the available functions within the category, then click a function to insert it at the cursor in the rules editor.

The function is inserted with placeholders for required parameters. You must replace the placeholders with valid parameter values. For example, if you use the function list to insert the ATTRN function into your rules, you see this:

```
ATTRN( «dimension», «element», «attribute» );
```

You must replace the <<dimension>>, <<element>>, and <<attribute>> placeholders to successfully return a numeric attribute value with ATTRN.

## Shortcut keys

There are many shortcut keys available in the rules editor. You can use these keys to edit statements, find and replace text, and navigate through the statements in the rules editor.

To see a full listing of shortcut key combinations, click , then click **Display shortcut keys**.

To hide the listing of shortcut keys, click , then click **Hide shortcut keys**.

## Validation and error highlighting

Some errors in rules are identified in real-time as you create a rule. Others are identified only when the rule is fully validated when you save the rule or click the **Validate business rules** icon .

Errors in your rules are indicated by an error symbol next to the line number in which the error occurs.

You can hover over either the error symbol to view a description of the error.

## Line wrapping

To enable or disable line wrapping in the editor, click .

## Text customization

To change the font and size of the text that is displayed in the rules editor, click , then select a font family and size.

## Locate the rule in the Databases tree

You can click the **Find in tree** button to immediately locate the rule in the Databases tree.

# Delete rules

You can delete business rules when you are a Modeler or an Administrator.

## Delete rules in Planning Analytics Workspace

### Procedure

1. Open Planning Analytics Workspace and log in with a user that has Modeler or Administrator permissions.
2. Open a workbench or a book. If in a book, enable Edit mode.
3. In the Databases tree, navigate to the database where you want to delete a rule, and expand the database to reveal the dimensions, cubes, and other associated object groups.
4. Expand the **Cubes** group, and the cube for which you want to delete rules.
5. Right-click the rule and select **Delete business rules**.

# Trace cell values

If a cell value is derived through consolidation or rules, you can review or 'trace' the underlying values that contribute to the consolidation or the rules that define the cell value.

### About this task

You can trace cell values only in cubes that reside on a IBM Planning Analytics TM1 database version 2.0.3 or later. If you are viewing a cube on a TM1 database prior to 2.0.3, the **Trace Cell** option is not available on the right-click menu from a cell.

The **Trace Cell** option is available only on cell values that are derived through consolidation or rules. The option is unavailable on leaf cells.

### Procedure

1. Right-click the cell you want to trace.
2. Click **Trace cell**.

   The **Trace cell** page displays a table with details about the cell you are tracing. The first row shows information about the original cell from which you initiated tracing. You can click the expand/collapse icon next to the cell coordinates to trace the original cell's value. As you trace the value, more rows appear to provide further details about how the original cell value is derived.

   The table includes the following columns:

   **Cell**
   : For the first row, identifies the original cell by its coordinates in the cube, by using a comma-separated list of member names. As you click the expand icon to trace the original value, more rows show the members that contribute to the original value.

   **Value**
   : The value of the original cell or member for the current table row. If the value is over 10,000, the value that is displayed is truncated to an integer, otherwise the value shows two decimal places. You can hover over the displayed value to view the actual raw value as it exists in the TM1 database. To enable the value to display, right-click a cell and select **Trace cell**. Then, click **Display unrounded values**.

   **Source**
   : Identifies the source of the value: Consolidation, Input (a leaf cell that is not rule calculated), or rule. If the source is a rule, the entire rule is displayed. If the rule is too long to appear in its entirety within the confines of the column, you can click the expand/collapse icon to reveal the full rule.
3. Click the expand/collapse icons in the **Cell** column to trace the original value as far as you want to go.

**Example**

| Cell | Value | Source |
|---|---|---|
| ⊖ Actual,Argentina,S Series 1.8 L Sedan,Units,2 Quarter | 746.00 | Consolidation |
| Apr | 285.00 | Input |
| May | 232.00 | Input |
| Jun | 229.00 | Input |

*Figure 3. Simple Consolidation*

In this example, the cell trace is initiated from a consolidation cell that is identified by the members Actual, Argentina, S Series 1.8 L Sedan, Units, 2 Quarter. When you click the expand icon in the first row of the Cell column, the three input leaf members and their values are revealed.

| Cell | Value | Source |
|---|---|---|
| ⊖ Actual,Argentina,S Series 2.0 L Sedan,Price,Jan | 27,390 | ⊖ ['Price']=N:DB('PriceCube',!actvsbud,!region,!model,!month);C:['Sales']\['Units']*1000; |
| PriceCube(Actual,Argentina,S Series 2.0 L Sedan,Jan) | 27,390 | Input |

*Figure 4. Simple Rule*

In this example, the cell trace is initiated from a cell that is derived through rules and identified by the members Actual, Argentina, S Series 2.0 L Sedan, Price, Jan. Because Price is calculated by a rule, the rule appears in the Source column. When you click the expand icon in the first row of the Cell column, the component of the rule that calculates the original cell value is revealed. In this case, the component is a reference to an Input value in the PriceCube.

| Cell | Value | Source |
|---|---|---|
| ⊖ Actual,Argentina,S Series 1.8 L Sedan,Gross Margin%,Mar | 55.75 | ['Gross Margin%']=['Gross Margin']\['Sales']*100; |
| ⊖ Gross Margin | 4,830.52 | Consolidation |
| ⊖ Sales | 8,664.49 | ['Sales']=N:['Price']*['Units']\1000; |
| ⊖ Price | 25,041 | ⊕ ['Price']=N:DB('PriceCube',!actvsbud,!region,!model,!month);C:… |
| PriceCube(Actual,Argentina,S Series 1.8 L Sedan,Mar) | 25,041 | Input |
| Units | 346.00 | Input |
| Variable Costs | 3,833.97 | Input |
| ⊖ Sales | 8,664.49 | ['Sales']=N:['Price']*['Units']\1000; |
| ⊖ Price | 25,041 | ⊖ ['Price']=N:DB('PriceCube',!actvsbud,!region,!model,!month); C:['Sales']\['Units']*1000; |
| PriceCube(Actual,Argentina,S Series 1.8 L Sedan,Mar) | 25,041 | Input |
| Units | 346.00 | Input |

*Figure 5. Combination*

In this example, the cell trace is initiated from a cell that is derived through rules. The cell is identified by the members Actual, Argentina, S Series 1.8 L Sedan, Gross Margin%, and Mar. Because Gross Margin% is calculated by a rule, the rule appears in the Source column.

Gross Margin% is calculated by using the members Gross Margin and Sales. When you click the expand icon in the first row of the Cell column, new rows for these members appear in the table. Gross Margin is a consolidation, while Sales is defined by a rule. You can click the expand icon next to both of these members to further trace the values that contribute to the original cell value.

## Check feeders

If a cube has a rule attached that uses SKIPCHECK and FEEDERS, you can use the **Check feeders** option to make sure that the members of consolidations are properly fed.

You can check feeders only from a consolidated cell; the **Check feeders** option is not available from leaf cells.

Follow the steps below to check feeders.

### Procedure

1. In an Exploration view, right-click the consolidated cell you want to check.
2. Select **Check feeders**.

   The **Check feeders** window opens, indicating whether the members of the selected consolidation are fed or unfed.

   If the members of the consolidation are unfed, you must edit the rule associated with the cube to contain FEEDERS statements that feed the members of the consolidation.

   For details on feeders, see Improving performance with feeders.

## Trace feeders

From an Exploration view, you can right-click a leaf cell and click **Trace feeders** to trace how a selected cell feeds other cells. You can use this capability to debug or improve the performance of your cube rules.

### About this task

You can feed only from a leaf member, so the **Trace feeders** option is not available for consolidated cells. However, the option is available for leaf cells defined by rules.

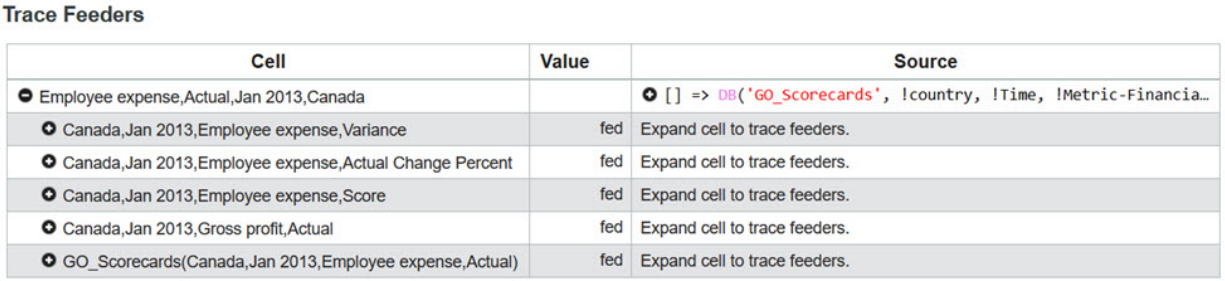
**Note:** You can trace feeders only from cubes that reside on Planning Analytics database version 2.0.3 or later. If your cube resides on a database before version 2.0.3, the **Trace feeders** option is not available when you right-click a leaf cell.

You can learn more about rules in general, and feeders in particular, in the TM1 Rules Guide.

### Procedure

1. In an Exploration view, right-click the leaf cell you want to trace.
2. Click **Trace Feeders**.

   The **Trace Feeders** window opens. When you open the window, the first row in the grid shows the cell from which you initiated feeder tracing. Following the first row you can see the cells that are directly fed by that cell.


Trace Feeders

| Cell | Value | Source |
|---|---|---|
| Employee expense,Actual,Jan 2013,Canada | | [] => DB('GO_Scorecards', !country, !Time, !Metric-Financia… |
| Canada,Jan 2013,Employee expense,Variance | fed | Expand cell to trace feeders. |
| Canada,Jan 2013,Employee expense,Actual Change Percent | fed | Expand cell to trace feeders. |
| Canada,Jan 2013,Employee expense,Score | fed | Expand cell to trace feeders. |
| Canada,Jan 2013,Gross profit,Actual | fed | Expand cell to trace feeders. |
| GO_Scorecards(Canada,Jan 2013,Employee expense,Actual) | fed | Expand cell to trace feeders. |

   The grid includes three columns: **Cell**, **Value**, and **Source**.

   **Cell**

   Identifies the cell by its coordinates in the cube using a comma-separated list of invariant names. If the cell is not from the cube in which feeder tracing was initiated, the cell is identified as `CubeName(Member0, Member1,...)`.

   An expand/collapse button appears next to the cell coordinates. You can click the button to reveal the cells that are fed by the current cell, and you can click the button again to hide the cells that are fed by the current cell.

**Value**
Indicates whether the cell is fed or is unfed.

**Source**

Displays the feeder statements that are associated with the cell. The statements are formatted and colorized as they appear in the rules editor.

To minimize space, feeders statements are initially shown truncated on a single line. An expand/collapse button appears next to the feeder statements. You can click the button to reveal all the feeder statements that are associated with the cell, and you can click the button again to collapse the statements to a single line.

When you first open the **Trace Feeders** window, the statements for the cells that are fed by the cell from which you initiated feeder tracing are unknown. The Source column displays the message "Expand cell to trace feeders." Click the expand/collapse button in the Cell column to calculate the feeder statements. If the cell is included in feeder statements, then the statements are displayed in the Source column and any fed cells appear as child entries. If the cell is not associated with any feeders, the Source column displays the message "This cell does not feed any cells." This is also the case as you trace through feeders. Calculation occurs when you click the expand/collapse button in the Cell column.

3. Click the expand/collapse button for any location in the Cell column to reveal how that cell feeds other cells in your model. If the cell feeds other cells, those other cells appear as children in the Cell column. You can continue to trace down through children until you reach the point where a child cell does not feed any other cells.

# TurboIntegrator processes

IBM Planning Analytics Workspace provides the ability to create, edit, and execute TurboIntegrator processes.

You can use TurboIntegrator processes to import or export data, create and maintain metadata/objects, and manage security on your database. TurboIntegrator processes must be created and maintained on a modeling workbench.

You can use the Planning Analytics Workspace process editor to edit processes that were created in other TM1 clients, such as Architect or Perspectives.

For complete details on TurboIntegrator processes, see the TM1 TurboIntegrator documentation. This documentation references a different user interface, which includes some capabilities that are not present in Planning Analytics Workspace. However, the general concepts, descriptions of process procedures, and examples are valid for both environments.

## Create and edit processes

You can create or edit a TurboIntegrator process in the process editor on a modeling workbench.

### Procedure

1. To create a new process:
   a) Open a modeling workbench.
   b) In the **Databases** tree, expand the database on which you want to create the new process.
   c) Right-click the **Processes** group.
   d) Click **Create process**.
   e) You can change the database where the process is stored by selecting a database from the **Database** list. The current database is selected by default.
   f) Enter a name for the process.
   g) Click **Create**.

2. To edit an existing process:
   a) Open a modeling workbench.
   b) Right-click the process in the **Databases** tree.
   c) Click **Edit process**.
3. Define a data source, set parameters, and enter your process statements in the process editor. Process statements are generally TM1 TurboIntegrator or Rules functions that execute actions upon data or metadata in your database.
   - For full descriptions of all TurboIntegrator functions, see TM1 TurboIntegrator Functions on IBM Knowledge Center.
   - For full descriptions of all Rules functions, see TM1 Rules Functions on IBM Knowledge Center. All Rules functions, with the exceptions of STET and ISLEAF, are valid in TurboIntegrator processes.

   A TurboIntegrator process has four distinct procedures that are executed sequentially when you run a process.

| Procedure | Description |
|---|---|
| Prolog | A series of statements to be executed before the data source is processed. |
| Metadata | A series of statements that update or create cube, dimensions, and other metadata structures during processing. |
| Data | A series of statements that manipulate values for each record in the data source. |
| Epilog | A series of statements to be executed after the data source is processed. |

   You must ensure that you create statements in the appropriate procedure within the process. For example, a statement that creates a new cube should be in the Metadata procedure. Similarly, any statements that update data values should be in the Data procedure.

   When you enter a statement within any procedure section in the process editor, it must be either before this line:

```
#****GENERATED STATEMENTS START****
```

   or after this line:

```
#****GENERATED STATEMENTS FINISH****
```

   Do not insert any of your statements between the generated statements start and finish lines, as anything between these lines can be overwritten.
4. Click **Save** to save the process.

## Define a data source

You can use the Planning Analytics Workspace process editor to import data from multiple data source types.

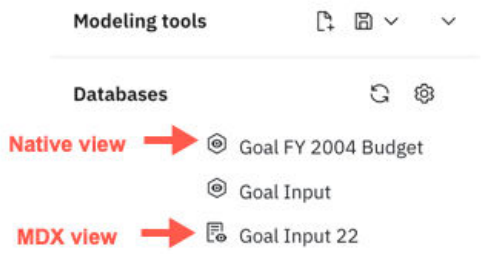
Each data source type requires specific steps to identify and define the data source.

## *Define a cube view data source*

The Planning Analytics Workspace process editor lets you define a cube view as a data source from which you can extract data and create or update objects and/or data.

*Define a cube view data source in Planning Analytics Workspace*

TurboIntegrator processes in Planning Analytics Workspace can use native or MDX views as data sources. You can differentiate a native view from an MDX view by the icon next to the view name in the data tree.




**Note:** MDX views that are saved as private views cannot be used as data sources for TurboIntegrator processes.

To define a view as a data source:

1. Open the process in the process editor.
2. Click the **Data Source** tab on the process editor.
3. Click the **Data source** menu, then click **Cube**.
4. Click the **Cube type** menu, then click the type of cube you want to use as a data source: **Cube** or **Control cube**.
5. Click the **Cube** menu (or **Control cube** menu if that's the type of cube you picked in the previous step), then click the cube you want to use as a data source.

   Alternatively, you can begin typing a cube name directly in the **Cube** box and the editor filters the list of available cubes that match the characters you enter.
6. Click the **View** menu, then click the specific view that you want to use as a data source.

   If there are no views available for the book, or if you want to create a new view or modify an existing view as a data source, click **Open view** to define view. After the view is saved, it is available in the **View** menu.

   **Important:** If you use a native view as a data source, when you modify the view with **Open view** and save it back to the server, it saves as an MDX view. Native views transform into MDX views when you open them in Planning Analytics Workspace. This important to note as variable runtime order in MDX views are based on the view and not on the order of dimensions as described in step 7.
7. Click **Preview** to view the variables in your data source.

   When you run a process that uses a native view as a data source, the view previews and reads variables in the order that the dimensions are listed on the cube. Any change to the dimensions in the view does not impact the order in which the variables are read.

   Alternatively, when you use an MDX view as a data source, the preview and variable runtime order depend on the view and not on the order of dimensions. This enables hierarchy support as the number of hierarchies in an MDX view can be different than the number of dimensions in the cube.
8. Click **Set variables** to initially set variable names and data types based on a review of the data source.

   Assigned variable names generally coincide with the dimension names in the data source. If the variable names that are assigned are not meaningful or familiar to you, you can click a variable name and type a new name. Variable names can contain only letters, numbers, and underscore and must start with a letter.

   **Note:** The variable order in an MDX view is Rows, Columns, and then Slicer (all left to right).

It's good practice to give the variables a meaningful name. Having meaningful names makes the process scripts easier to read and troubleshoot.

Similarly, if the data type for any variable is misidentified, you can click the data type and pick a different type (**Numeric**, **String**, or **Ignore**). Choose the **Ignore** data type if you want the variable to be ignored and just kept as a placeholder.

9. Click **Save** to save the process.

### *Define a dimension set data source*

Follow these steps to define a dimension set as a data source for a process.

*Define a dimension set data source in Planning Analytics Workspace*

## Procedure

1. Open the process in the process editor.
2. Click the **Data Source** tab on the process editor.
3. Click the **Data source** menu, then click **Dimension**.
4. Click the **Dimension type** menu, then click the type of dimension you want to use as a data source: **Dimension** or **Control dimension**.
5. Click the **Dimension** menu (or **Control dimension** menu if that's the type of dimension you picked in the previous step), then click the dimension you want to use as a data source. Alternatively, you can begin typing a dimension name directly in the **Dimension** box and the editor filters the list of available dimension that match the characters you enter.
6. Click the **Hierarchy** menu, then click the hierarchy you want to use as a data source. You can also begin typing an hierarchy name directly in the **Hierarchy** box and the editor filters the list of available dimension that match the characters you enter.
7. Click the **Set** menu, then click the set you want to use as a data source.
8. Click **Preview** to view the variables in your data source.
9. Click **Set variables** to initially set variable names and types based on a review of the data source.

   Assigned variable names generally coincide with the dimension names in the data source. If the variable names that are assigned are not meaningful or familiar to you, you can click a variable name and type a new name. Variable names can contain only letters, numbers, and underscore and must start with a letter.

   It's good practice to give the variables a meaningful name. Having meaningful names makes the process scripts easier to read and troubleshoot.

   Similarly, if the data type for any variable is misidentified, you can click the data type and pick a different type.

10. Click **Save**.

### *Define a file data source*

You can define a text file as a data source for a process.

## About this task

The file you use as a data source must be a delimited text file. It can have any file extension.

Microsoft Excel (.xls/.xlsx) files cannot be used as a data source, nor can fixed-width text files.

*Define a file data source in Planning Analytics Workspace*

## Procedure

1. Open the process in the process editor.
2. Click the **Data Source** tab on the process editor.

3. Click the **Data source** menu, then click **File**.
4. Click the **File** menu, then click the file you want to use as a data source.

   If the file you want to use as a data source is not available in the list , click **Upload file** and upload the file you want to use. After the file is uploaded, it is available in the **File** menu.

   The process editor identifies the delimiter character, as well as the quote character, decimal separator, and thousand separator, and number of header records in your source. If the process editor incorrectly identifies any of these items, you can modify the settings.

   **Important:** Your data source file might contain one header record, multiple header records, or zero header records. You need to know how many header records your source contains and set the **Header records** accordingly to ensure that all data in the source file is processed correctly.
5. Click **Preview** to view the variables in your data source.

   You can click **View more** if you want to review additional rows in your data source.
6. Click **Set variables** to initially set variable names and types based on a review of the data source.

   If the variable names that are assigned are not meaningful or familiar to you, you can click a variable name and type a new name. Variable names can contain only letters, numbers, and underscore and must start with a letter.

   It's good practice to give the variables a meaningful name. Having meaningful names makes the process scripts easier to read and troubleshoot.

   Similarly, if the data type for any variable is misidentified, you can click the data type and pick a different type.
7. Click **Save** to save the process.

### *Define a location data source*

You can define a local or network location as a data source for a process.

## About this task

When specifying a location data source for a process on a Planning Analytics database, the location must satisfy these requirements:

- For Planning Analytics on Cloud or Planning Analytics Local, a path relative to the Planing Analytics data directory. For example, `./data_sources/filename.txt` or `../data_sources/filename.txt`
- For Planning Analytics on Cloud, a path to the shared `s:/` directory that is available in all Planning Analytics on Cloud environments. For example, `s:/data_sources/filename.txt`
- For Planning Analytics Local, you can use a path to a mapped network drive, a local drive, or a UNC path if the Planning Analytics database is running on a service account that has access to the location.

When specifying a location data source for a process on TM1 Database 12 in Planning Analytics as a Service and Planning Analytics on Cloud Pak for Data, the location must be specified as a URL using either the HTTP or HTTPS protocol. The URL must be accessible from the Planning Analytics Engine.

There is no interface for providing authentication credentials, but if the URL you use for the location data source does require authentication, you can authenticate by including an ExecuteHttpRequest function in the Prolog of your process script that identifies the URL and includes an Authentication Header. Functions in the Prolog are executed before the data source is loaded. The ExecuteHttpRequest retrieves a cookie that is stored for the scope of the process and is used for authentication when the process loads the data source.

For example:

```
#Section Prolog
ExecuteHttpRequest ('POST', 'https://location_data_source_URL', '-h authentication_header');
```

*Define a location data source in Planning Analytics Workspace*

## Procedure

1. Open the process in the process editor.
2. Click the **Data Source** tab on the process editor.
3. Click the **Data source** menu, then click **Location**.
4. Click **Set variables** to initially set variable names and types based on a review of the data source.

   If the variable names that are assigned are not meaningful or familiar to you, you can click a variable name and type a new name. Variable names can contain only letters, numbers, and underscore and must start with a letter.

   It's good practice to give the variables a meaningful name. Having meaningful names makes the process scripts easier to read and troubleshoot.

   Similarly, if the data type for any variable is misidentified, you can click the data type and pick a different type.
5. Click **Save** to save the process.

### *Define a database connection data source*

You can define a database connection as a data source for a process.

*Define a database connection data source in Planning Analytics Workspace*

## About this task

To use a database connection as a data source for a process, the following conditions must be met:

- For both Planning Analytics Workspace on cloud and Planning Analytics Workspace Local, you must be using IBM Planning Analytics version 2.0.4 or later.
- For Planning Analytics Workspace on cloud, if the database resides anywhere other than the computer on which your TM1 database resides, you must create an IBM Satellite Connector connection to access the data.
- For Planning Analytics Workspace Local, the client software for your relational database must be running on the same computer on which your TM1 database resides. For example, an ODBC Excel data source cannot be located on a mapped network drive; it must exist on the computer where the TM1 database resides.
- Regardless whether you are working with Planning Analytics Workspace on cloud or Planning Analytics Workspace Local, the ODBC data source must already be established before you can use it as a process data source.

## Procedure

1. Open the process in the process editor.
2. Click the **Data Source** tab on the process editor.
3. Click the **Data source** menu, then click **Database connection**.
4. If required, enter a **Username** and **Password** to connect to the database.

   Password length is constrained only by the buffer limit, which is 2499 characters.
5. Enter a **Database connection query** to extract data from the source. The syntax and format of the query will vary depending on which type of database you are accessing.

   If the query references a table name that contains spaces, you must enclose the name in double quotes.
6. Click **Preview** to view the variables in your data source.

   You can click **View more** if you want to review additional rows in your data source.

7. Click **Set variables** to initially set variable names and types based on a review of the data source.

   If the variable names that are assigned are not meaningful or familiar to you, you can click a variable name and type a new name. Variable names can contain only letters, numbers, and underscore and must start with a letter.

   It's good practice to give the variables a meaningful name. Having meaningful names makes the process scripts easier to read and troubleshoot.

   Similarly, if the data type for any variable is misidentified, you can click the data type and pick a different type.
8. Click **Save** to save the process.

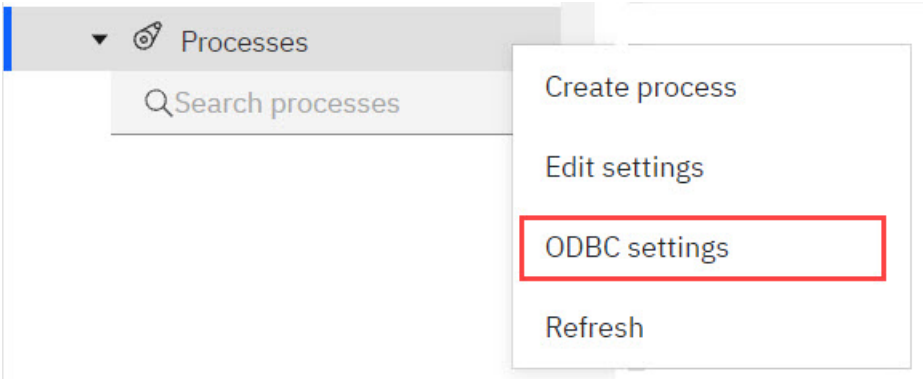
*Modify connection configuration for multiple processes that use ODBC data sources*
Each process with an ODBC data source includes the declaration of the ODBC connection, username, and password as part of the data source connection configuration. When your database contains multiple processes that use the same data source connection configuration, it can be laborious to update each process individually when the connection configuration changes. Use the connection manager to modify the ODBC connections for multiple processes simultaneously.

## About this task

The connection manager is available only on the modeling workbench. You can't use the connection manager in a modeling book.

## Procedure

1. To open the connection manager, click the **Actions** button ⋮ on the Process node in the Databases tree, then click **ODBC settings**.




2. In the connection manager, select the processes for which you want to modify connection configuration.
3. Click **Edit**.
4. Select the connection parameters you want to change and specify new parameter values.

   You can modify any combination of **Database connection name** (connection type), **Username**, and **Password**.

   You can remove a username or password from a connection by selecting the **Change username** or **Change password** options and leaving the parameter value empty.
5. Click **Apply**.

## Results

After you apply the connection changes, the modified connection configuration is applied and saved to each of the processes you selected.

A status message informs you of the success or failure of the connection configuration update. If the update is not successful for all processes, you'll see a list of the processes that were not updated.

### *Define a JSON data source (TM1 Database 12 only)*

You can define a JSON file as a data source for a process.

## About this task

You can use a JSON file as a data source only when using Planning Analytics Workspace connected to TM1 Database 12.

TM1 Database 12 is the next-generation Planning Analytics database, available on the IBM Cloud Pak® for Data and IBM Planning Analytics as a Service platforms.

All values passed to string variables are JSON values. This is true when the entire record of values are passed onto one variable, and when individual values are assigned to individual variables using variable mapping. Strings use JSON notation, and are returned enclosed in double quotations.

If a JSON value is mapped to a numeric variable, an implicit conversion is attempted. If the JSON value is a numeric value, then that value is returned. If the JSON value is a boolean, a 1 is returned if the boolean is true, and a 0 is returned if the boolean is false. If the value is a null, then a value of 0 is returned, and the 'is null in database' flag is set to true.

When using the REST API, the JSON data source can be configured by specifying `JSON` as the name for the type of data source being consumed by the process. The`dataSourceNameForServer` property specifies the JSON resource. Optionally, you can specify a `jsonRootPointer` or `jsonVariableMapping` property to provide additional details of how the JSON should be interpreted and the data mapped to variables in the Turbo Integrator.

## Procedure

1. Open the process editor.
2. Click the **Data Source** tab on the process editor.
3. Click the **Data source** menu, then click **JSON file**.
4. Click **Select file**

   If the file you want to use as a data source is not available in the list, click **Upload file** and upload the file you want to use. After the file is uploaded, it is available for selection as a data source in File manager.

   **Tip:** To preview the content and structure of a file, click the **Options** menu ⋮ for the file, then click **Preview**.
5. Select the file you want to use as a data source, then click **Select**.
6. Enter the **JSON root** within the JSON file to use as your data source. The JSON root must be an array element.
7. Click **Preview** to view the variables in your data source.
8. Click **Set variables** to initially set variable names and types based on a review of the data source.

   If the variable names that are assigned are not meaningful or familiar to you, you can click a variable name and type a new name. Variable names can contain only letters, numbers, and underscore and must start with a letter.

   It's good practice to give the variables a meaningful name. Having meaningful names makes the process scripts easier to read and troubleshoot.

   Similarly, if the data type for any variable is misidentified, you can click the type and pick a different type.
9. Click **Save** to save the process.

## Add variables to a process

The ability to add variables to a process is useful when you are creating a process, but the data source is not available. It's also useful when the number of columns in the data source changes, as can occur when the data source of a process is modified at run time.

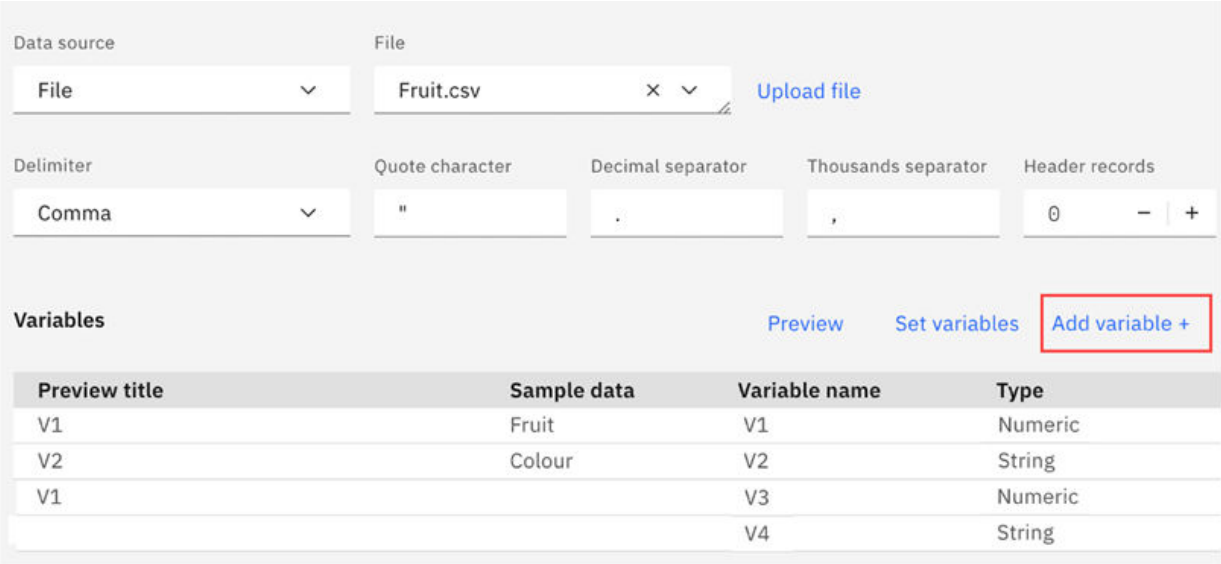
### About this task

A variable name cannot be empty and must adhere to the Planning Analytics naming conventions.

If you have more variables in a process than columns in the data source, the process still works. The extra variables will simply be empty.

### Procedure

1. Open the process in the process editor.
2. Click **Add variable**.




3. Enter a name for the variable.

   You can also delete any variable in a process. Hover the cursor on any variable name to reveal, and then click, the icon.

## Allow a process to modify security data

By default, security access is disabled when you create a new process. This means that the process cannot modify security data. If you want to allow a process to modify security, you must enable security access for the process.

### *Allow a process to modify security data in Planning Analytics Workspace*

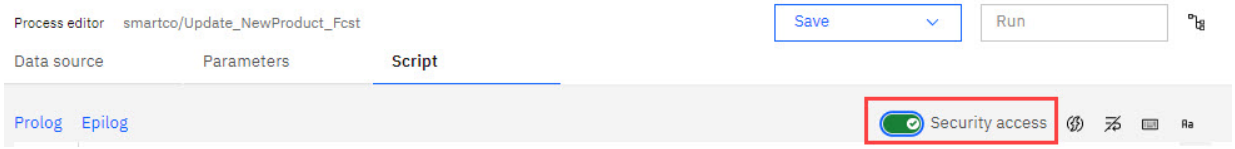
### About this task

You must have an Administrator or Modeler role in Planning Analytics Workspace and be a member of either the ADMIN or SecurityAdmin group in TM1 to enable security access. When security access is enabled, only Planning Analytics Workspace users who are members of TM1 ADMIN group can modify the process.

You can enable or disable security access from the Databases tree on a workbench. Right-click a process in the tree, then click **Enable security access** or **Disable security access**. You can also manage security access directly in the process editor.

## Procedure

1. Open the process in the process editor.
2. Click the **Script** tab.
3. Enable or disable the **Security access** option.




# Create and edit process parameters

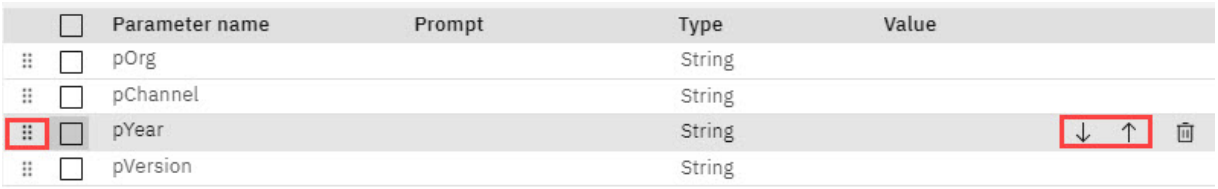
A TurboIntegrator process can include runtime parameters that are passed into the process upon execution. Parameter values are set when a user runs the process and answers prompts for parameter values.

## *Create and edit process parameters in Planning Analytics Workspace*

## Procedure

1. Open the process in the process editor.
2. Click the **Parameters** tab on the process editor to open the parameters view.
3. Click **Add parameter**.

   A new parameter is added with a default name, type, and value. You can modify all these defaults.
4. In the **Name** cell, enter a parameter name. Parameter names should adhere to "Naming conventions" on page 1264, and should not include any TurboIntegrator reserved words.
5. In the **Prompt** cell, enter the question or prompt that a user will respond to when supplying a parameter value.
6. In the **Type** cell, choose a parameter type: numeric or string.
7. In the **Value** cell, enter a default value for the parameter.
8. Click **Add parameter** to create additional parameters, then repeat steps 2 through 5.
9. To change the order of parameters, select a parameter, then use the arrow buttons to move the parameter up or down in the process editor. You can also click a parameter handle and drag the parameter to a new position.




10. To delete a parameter, select the parameter, then click .

## Results

When a user runs the process, a dialog box requests answers to the prompts that you defined in step 4. The answers that the user supplies are passed into the process as parameter values.

# Insert a code snippet into a process

The process editor in Planning Analytics Workspace provides predefined code snippets for repetitive tasks that can be used with TurboIntegrator processes.

## About this task

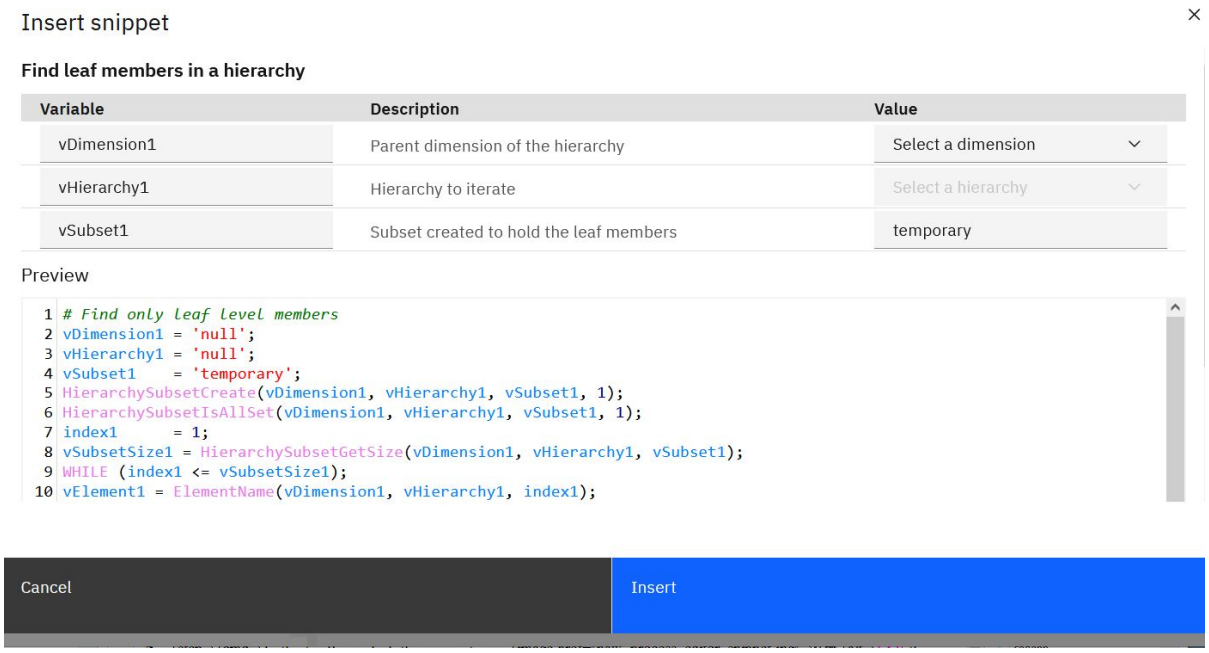
When running TurboIntegrator processes, users often repeat the same tasks, changing only the variable or object names. The code snippets simplify the repetitive tasks, making the process less error-prone.

## Procedure

1. Open the process in the process editor.
2. Click the **Script** tab.
3. In the toolbar, click the snippet icon .

   Click the category for the type of snippet you want to insert, such as **Create**, **Iterators**, **Other**, or **Custom**.
4. Click the snippet you want to insert into the process. If you are inserting a Custom snippet, you must click **Select** after you click the snippet.

   For example, from the **Iterators** category, click the snippet **Find leaf members in a hierarchy**.




5. Specify values in the **Variable** and **Value** columns.

   As you specify values in these columns, the **Preview** is updated with your input.
6. Click **Insert**.

   The code snippet is inserted in the line under the current position of the cursor.
7. Save the process.

## Create and upload a custom snippet

Create a custom process snippet to save a frequently-used process script as a `.json` file, which can be inserted into any process on your database.

### About this task

A custom snippet is composed of several objects or properties, some required and others optional, all using standard JSON syntax. The objects and properties are described in Table 1.

*Table 11. Snippet objects and properties*

| Snippet component | Required? | Description |
|---|---|---|
| **snippetId** | Required | A unique identifier for the snippet.<br>Example: `"snippetId": "ITERATION_SNIPPET",` |
| **name** | Required | A name for the snippet.<br>Example: `"name": "Iterate over members in a hierarchy",` |
| **description** | Required | A description of the snippet. Though this is a required property, it can be empty if you don't want to provide a description.<br>Example: `"description": "This snippet iterates over members in a hierarchy and performs an action on each leaf member",` |
| **auxliaryVariables** | Optional | Auxiliary variables must be included and defined in the snippet only if your script includes auxiliary variables. These variables are used in the script during runtime. For example, a counter variable for a WHILE loop.<br>Auxiliary variables are specified as an array of the auxiliary variable names that are present in the **script**.<br>Example:<br>`"auxiliaryVariables": [ "index", "vElement" ]` |

*Table 11. Snippet objects and properties (continued)*

| Snippet component | Required? | Description |
|---|---|---|
| **variables** | Optional | Variables must be included and defined in the snippet only if your **script** includes variables. These are the variables that must be initialized before inserting the snippet into a process.<br><br>Variables are specified as an array, using the variable object format described later in this documentation.<br><br>Example:<br><br>`"variables": [ { "variableName": "vDimension", "description": "Parent dimension of the hierarchy", "valueDefinition": { "type": "DIM", "defaultVal": null } }, { "variableName": "vHierarchy", "description": "Hierarchy to iterate", "valueDefinition": { "parent": "vDimension", "type": "DIM_HIER", "defaultVal": null } }],` |
| **script** | Required | An array of strings containing the process code. Each string value will be a new line when the snippet is inserted in the process editor.<br><br>In the script, variables are referenced using the format `vName:`*`AssignedVariableName`*, where *`AssignedVariableName`* is the name you assign with the **variableName** property.<br><br>In the script, variable values are referenced using the format `vValue:`*`AssignedVariableName`*<br><br>Auxiliary variables are referenced using the format `aux:`*`AuxiliaryVariableName`*, where *`AuxiliaryVariableName`* is an auxiliary variable name that has been defined in the **auxliaryVariables** property.<br><br>Example:<br><br>`"script": [ "script line 1;", "script line 2;", "script line 3", ]` |

If your `.json` file includes **variables**, they must be specified using this object format:

```
"variables": [
            {
           "variableName": "name",
           "description": "Brief description",
           "valueDefinition": {
               "parent": "parentOfHierarchy",
               "type": "aValidTye",
```

```
            "defaultVal": "value"
        }
    }],
```

*Table 12. Variable object components*

| Variable component | Required? | Description |
|---|---|---|
| **variableName** | Required | The name of the variable.<br>Example:<br>`"variableName": "vDimension"`<br>Once the variable name is defined, it is referenced in the **script** using the format `vName:`*`AssignedVariableName`*. For example, after defining `"variableName": "vDimension"`, you would reference the variable as `vName:vDimension` in the **script** object. |
| **description** | Optional | A brief description of the variable.<br>Example:<br>`"description": "Parent dimension of the hierarchy"` |
| **valueDefinition** | Required | A value definition object containing these properties:<br>**parent**<br>The parent dimension of the hierarchy that is referenced by the variable. This property is required only if the **type** property is `DIM_HIER`.<br>**type**<br>The type of value that the variable references. Valid types are:<br>• `CUB` - a cube<br>• `DIM` - a dimension<br>• `DIM_HIER` - a hierarchy<br>• `STRING` - a string<br>• `NUMERIC` - a numeric value<br>**defaultVal**<br>Optionally, a default value for the variable.<br>Example:<br>`"valueDefinition": { "parent": "vDimension", "type": "DIM_HIER", "defaultVal": null }` |

## Procedure

1. Create a new `.json` file.
2. Create your custom snippet using the components described in this topic.
3. Save the snippet.
4. Open a modeling workbench.

5. On the **Databases** tree, expand the database where you want to upload your custom snippet.
6. Open an existing process in the process editor or right-click the **Processes** node, then click **Create process** to open a new process editor.
7. Click the **Script** tab on the process editor.
8. Click the snippet icon  , then click **Custom**.
9. On the **Snippet manager** page, click **Upload snippets**.
10. Drag and drop or navigate to the `.json` file that contains your custom snippet, then click **Upload**.

    The custom snippet you uploaded is now available for reuse at any time.
11. Click **Close** and then **Cancel** to dismiss the **Snippet upload** and **Snippet manager** pages, respectively.

    For details on inserting a custom snippet into a process, see “Insert a code snippet into a process” on page 810.

### Example

This example presents a custom snippet that iterates over members in a hierarchy and performs an action on each 0-level member encountered in the hierarchy. The snippet writes member names to a file using the `AsciiOutput` function, but you could modify the code to perform whatever action you want within the WHILE loop.

To further your understanding of custom snippets, you can work through a complete tutorial to create this exact snippet.

```
{
  "snippetId": "ITERATOR_OVER_MEMBERS_IN_A_HIERARCHY",
  "name": "Iterate over members in a hierarchy",
  "description": "This snippet iterates over members in a hierarchy and performs an action on
each leaf member",
  "auxiliaryVariables": [
        "vSubsetSize",
        "index",
        "vElement"
    ],
    "variables": [
        {
            "variableName": "vDimension",
            "description": "Parent dimension of the hierarchy",
            "valueDefinition": {
                "type": "DIM",
                "defaultVal": null
            }
        },
        {
            "variableName": "vHierarchy",
            "description": "Hierarchy to iterate",
            "valueDefinition": {
                "parent": "vDimension",
                "type": "DIM_HIER",
                "defaultVal": null
            }
        },
        {
            "variableName": "vSubset",
            "description": "Subset created to hold all members ",
            "valueDefinition": {
                "type": "STRING",
                "defaultVal": "temporary"
            }
        }],
    "script": [
            "# Iterate over members in a hierarchy",
            "vName:vDimension = 'vValue:vDimension';",
            "vName:vHierarchy = 'vValue:vHierarchy';",
            "vName:vSubset = 'vValue:vSubset';",
            "HierarchySubsetCreate(vName:vDimension, vName:vHierarchy, vName:vSubset, 1);",
            "HierarchySubsetIsAllSet(vName:vDimension, vName:vHierarchy, vName:vSubset, 1);",
            "aux:index = 1;",
            "aux:vSubsetSize = HierarchySubsetGetSize(vName:vDimension, vName:vHierarchy,
vName:vSubset);",
```

```
            "WHILE (aux:index <= aux:vSubsetSize);",
            "   aux:vElement = ElementName(vName:vDimension, vName:vHierarchy, aux:index);",
            "   IF ( ElementLevel(vName:vDimension, vName:vHierarchy, aux:vElement) = 0);",
            "       # Do something with vElement such as a write it to a file;",
            "       AsciiOutput('model_upload/snippet.txt', aux:vElement);",
            "   ENDIF;",
            "   aux:index = aux:index + 1;",
            "   aux:vElement = ElementName(vName:vDimension, vName:vHierarchy, aux:index);",
            "END;"
      ]
}
```

## Manage custom snippets

Use the **Snippet manager** to upload new custom snippets, download existing custom snippets, delete snippets, or create folders to organize your custom snippets.

### Procedure

1. On the **Databases** tree, expand the database where you want to manage your custom snippets.
2. Open an existing process in the process editor or right-click the **Processes** node, then click **Create process** to open a new process editor.
3. Click the **Script** tab on the process editor.
4. Click the snippet icon  , then click **Custom** to open the **Snippet manager** .

   You can click a column header to sort snippets by **Name** or **Last updated** date.
5. To upload a custom snippet, click the **Upload snippets** icon, then drag and drop or navigate to the `.json` file that contains your custom snippet and click **Upload**.
6. To add a new folder, click the **Create new folder** icon.
7. To download a snippet, select the snippet then click **Download**.
8. To delete one or more snippets, select the snippets then click **Delete**.

## Try it out: Custom snippets tutorial

Work through this tutorial to understand the process of converting an existing process script to a reusable custom snippet.

### About this task

This tutorial converts this example process script into a custom snippet.

```
# Iterate over members in a hierarchy
vDimension = 'null';
vHierarchy = 'null';
vSubset = 'temporary';
HierarchySubsetCreate(vDimension, vHierarchy, vSubset, 1);
HierarchySubsetIsAllSet(vDimension, vHierarchy, vSubset, 1);
index = 1;
vSubsetSize = HierarchySubsetGetSize(vDimension, vHierarchy, vSubset);
WHILE (index <= vSubsetSize);
   vElement = ElementName(vDimension, vHierarchy, index);
   IF ( ElementLevel(vDimension, vHierarchy, vElement) = 0);
       # Do something with vElement such as a write it to a file
       AsciiOutput('model_upload/snippet.txt', vElement);
   ENDIF;
   index = index + 1;
   vElement = ElementName(vDimension, vHierarchy, index);
END;
```

The snippet you'll create in this tutorial prompts for variable values and then inserts the resulting script into a process. The script iterates over all members in the specified hierarchy and performs an action on each member. In this case, the script uses the `AsciiOutputFunction` to write each 0-level (leaf) member to the `snippet.txt` file. You can insert the custom snippet that you create in this tutorial into any new or existing process on your database and run it against any existing hierarchy.

## Procedure

1. Create a new JSON file named `IterateOverMembersInHierarchy.json`. The file extension must be .json, as this is the only file extension recognized for custom snippets.
2. Identify variables and auxiliary variables in the process script.

   This is the most important factor when converting a script into a snippet. Variables must be initialized before the snippet is executed, while auxiliary variables are required at run time.

   In the example script, you can identify three variables that must be initialized before the snippet is executed: *vDimension*, *vHierarchy*, and *vSubset*. The three auxiliary variables are *index*, *vSubsetSize*, and *vElement*. *vSubsetSize* and *vElement* store values needed during the runtime of the script.
3. To start building your custom snippet, define a **`snippetID`**, **`name`**, and **`description`** for the snippet in the `IterateOverMembersInHierarchy.json` file.

```
{
  "snippetId": "ITERATOR_OVER_MEMBERS_IN_A_HIERARCHY",
  "name": "Iterate over members in a hierarchy",
  "description": "This snippet iterates over members in a hierarchy and performs an action
on each leaf member"
}
```

   The name and description provide information when a user inserts the custom snippet into a process.
4. Add the three auxiliary variables identified in step 2: *index*, *vSubsetSize*, and *vElement*.

```
{
  "snippetId": "ITERATOR_OVER_MEMBERS_IN_A_HIERARCHY",
  "name": "Iterate over members in a hierarchy",
  "description": "This snippet iterates over members in a hierarchy and performs an action
on each leaf member"

  "auxiliaryVariables": [
        "index",
        "vSubsetSize",
        "vElement"
    ],
}
```

5. Add the three variables identified in step 2: *vDimension*, *vHierarchy*, and *vSubset*.

   For each variable, specify a **`variableName`**, **`description`**, and **`valueDefinition`**.

   **`valueDefinition`** requires that you define both a **`type`** and a **`defaultVal`**.

   - *vDimension* is a dimension, so **`type`** is `DIM`. The **`defaultVal`** is `null`, which means that you select the dimension when you insert the snippet into a process.
   - *vHierarchy* is a hierarchy so **`type`** is `DIM_HIER`. The **`defaultVal`** is `null`, which means that you select the hierarchy when you insert the snippet into a process.
   - *vSubset* is a subset so **`type`** is `STRING`. The **`defaultVal`** is `temporary`, but this subset name can be edited when you insert the snippet into a process.

```
{
  "snippetId": "ITERATOR_OVER_MEMBERS_IN_A_HIERARCHY",
  "name": "Iterate over members in a hierarchy",
  "description": "This snippet iterates over members in a hierarchy and performs an action
on each leaf member"

  "auxiliaryVariables": [
        "index",
        "vSubsetSize",
        "vElement"
    ],

   "variables": [
        {
            "variableName": "vDimension",
            "description": "Parent dimension of the hierarchy",
            "valueDefinition": {
                "type": "DIM",
                "defaultVal": null
```

```
            }
        },
        {
            "variableName": "vHierarchy",
            "description": "Hierarchy to iterate",
            "valueDefinition": {
                "parent": "vDimension",
                "type": "DIM_HIER",
                "defaultVal": null
            }
        },
        {
            "variableName": "vSubset",
            "description": "Subset created to hold all members ",
            "valueDefinition": {
                "type": "STRING",
                "defaultVal": "temporary"
            }
        }],
}
```

6. Create a **script** property in your .json file.

```
{
  "snippetId": "ITERATOR_OVER_MEMBERS_IN_A_HIERARCHY",
  "name": "Iterate over members in a hierarchy",
  "description": "This snippet iterates over members in a hierarchy and performs an action
on each leaf member"

  "auxiliaryVariables": [
        "index",
        "vSubsetSize",
        "vElement"
    ],

   "variables": [
        {
            "variableName": "vDimension",
            "description": "Parent dimension of the hierarchy",
            "valueDefinition": {
                "type": "DIM",
                "defaultVal": null
            }
        },
        {
            "variableName": "vHierarchy",
            "description": "Hierarchy to iterate",
            "valueDefinition": {
                "parent": "vDimension",
                "type": "DIM_HIER",
                "defaultVal": null
            }
        },
        {
            "variableName": "vSubset",
            "description": "Subset created to hold all members ",
            "valueDefinition": {
                "type": "STRING",
                "defaultVal": "temporary"
            }
        }],

   "script": [ ]
}
```

7. Copy the entirety of the example process script and paste it into the **script** property in the snippet.
8. Reformat the code in the **script** property into an array of comma-separated strings, with each line an individual string.

   For example, the first few lines of code in the **script** property should look like this

```
"# Iterate over members in a hierarchy",
"vDimension = 'null';",
"vHierarchy = 'null';",
"vSubset = 'temporary';",
```

   When you insert the custom snippet into a process, each string is a new line in the process editor.

9. Edit the code in the **script** property by updating all variable names using the format *vName:VariableName* and all initialized values using the format *vValue:VariableName*..

   For example, `vDimension = 'null';` becomes `vName:vDimension = 'vValue:vDimension';`.
10. Update all auxiliary variables in the **script** property using the format *aux:AuxiliaryVariableName*.

    For example, `index = 1;` becomes `aux:index = 1;`.

    The contents of the `IterateOverMembersInHierarchy.json` file should now look like this:

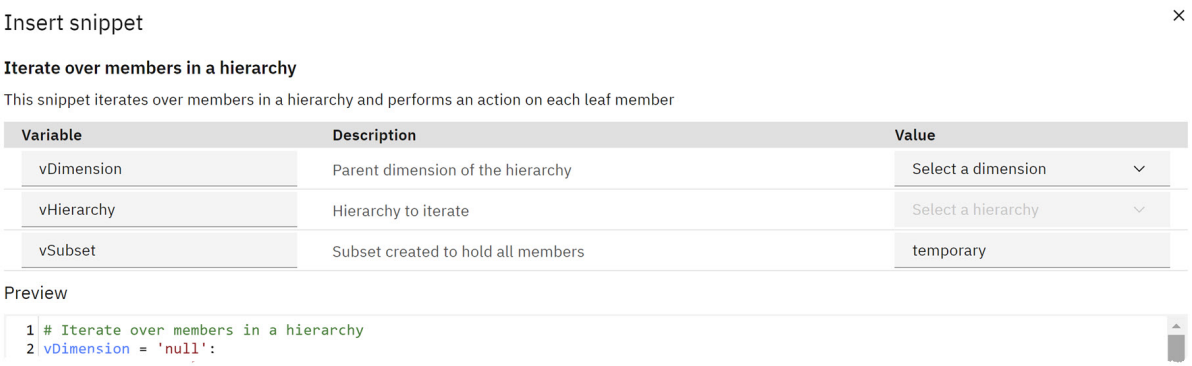
```
{
  "snippetId": "ITERATOR_OVER_MEMBERS_IN_A_HIERARCHY",
  "name": "Iterate over members in a hierarchy",
  "description": "This snippet iterates over members in a hierarchy and performs an action
on each leaf member",
  "auxiliaryVariables": [
        "vSubsetSize",
        "index",
        "vElement"
    ],
    "variables": [
        {
            "variableName": "vDimension",
            "description": "Parent dimension of the hierarchy",
            "valueDefinition": {
                "type": "DIM",
                "defaultVal": null
            }
        },
        {
            "variableName": "vHierarchy",
            "description": "Hierarchy to iterate",
            "valueDefinition": {
                "parent": "vDimension",
                "type": "DIM_HIER",
                "defaultVal": null
            }
        },
        {
            "variableName": "vSubset",
            "description": "Subset created to hold all members ",
            "valueDefinition": {
                "type": "STRING",
                "defaultVal": "temporary"
            }
        }],
    "script": [
            "# Iterate over members in a hierarchy",
            "vName:vDimension = 'vValue:vDimension';",
            "vName:vHierarchy = 'vValue:vHierarchy';",
            "vName:vSubset = 'vValue:vSubset';",
            "HierarchySubsetCreate(vName:vDimension, vName:vHierarchy, vName:vSubset, 1);",
            "HierarchySubsetIsAllSet(vName:vDimension, vName:vHierarchy, vName:vSubset,
1);",
            "aux:index = 1;",
            "aux:vSubsetSize = HierarchySubsetGetSize(vName:vDimension, vName:vHierarchy,
vName:vSubset);",
            "WHILE (aux:index <= aux:vSubsetSize);",
            "   aux:vElement = ElementName(vName:vDimension, vName:vHierarchy, aux:index);",
            "   IF ( ElementLevel(vName:vDimension, vName:vHierarchy, aux:vElement) = 0);",
            "       # Do something with vElement such as a write it to a file;",
            "       AsciiOutput('model_upload/snippet.txt', aux:vElement);",
            "   ENDIF;",
            "   aux:index = aux:index + 1;",
            "   aux:vElement = ElementName(vName:vDimension, vName:vHierarchy, aux:index);",
            "END;"
      ]
}
```

11. Create a new process on your database without a data source.
12. Click the **Script** tab on the process editor.
13. On the toolbar, click the snippet icon  , then click **Custom**.
14. On the **Snippet manager** page, click **Upload snippets**.
15. Specify your `IterateOverMembersInHierarchy.json` file, then click **Upload**.

The custom snippet you just created and uploaded is now available for reuse at any time.

16. Click **Close** and then **Cancel** to dismiss the **Snippet upload** and **Snippet manager** pages, respectively.
17. On the **Script** tab of the process editor, position the cursor between the `#Section prolog` and `#Section epilog`.
18. Click the snippet icon, then click **Custom**.
19. Click `IterateOverMembersInHierarchy.json` file, then click **Select**.

    The **Insert snippet** page opens so you can set variable values and preview the process code that will be inserted by the snippet.




20. Click **Select a dimension** and pick the dimension you want use.
21. Click **Select a hierarchy** and pick the hierarchy you want to iterate.

    You can leave the *vSubset* value as `temporary`. The code creates a temporary subset that persists only for the duration of the process execution.
22. Click **Insert**.

    Script based on the custom snippet is inserted into the process, using the variable values you specified on the **Insert snippet** page




23. Click **Save** to save the process.
24. Click **Run**.

## Results

When you run the process, the process creates a temporary subset and iterates over each member. When the process encounters a 0-level member, it writes the member name to the `snippet.txt` file in the Planning Analytics Workspace file manager.

To review the file, right-click your database in the **Databases** tree on a modeling workbench, then click **File manager**. Select **snippet.txt**, then click **Download**. The file should contain all the 0-level members of the hierarchy you selected when you inserted the custom snippet into the process.

# Use the process editor

The Planning Analytics Workspace process editor includes several features that simplify the creation and validation of TurboIntegrator processes.

## Use the process editor in Planning Analytics Workspace

These features help you create and manage processes in the Planning Analytics Workspace process editor.

### Open the process editor in a separate window

To open an the editor in a separate window, click **Open tab in floating window**.

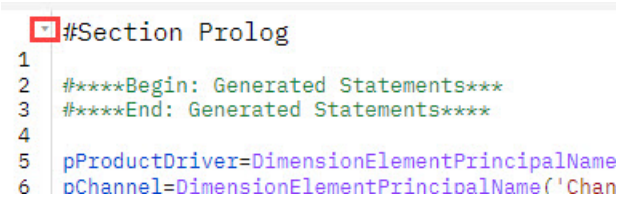
For more information, see "Open editors in a separate window" on page 730.

### Procedure links

A TurboIntegrator process contains four procedures: Prolog, Metadata, Data, and Epilog. The Script tab on the process editor includes links to each procedure so you can quickly jump to the procedure you want to edit.

### Procedure section collapse/expand

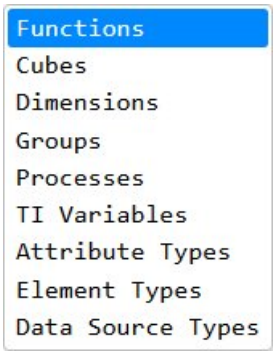
Click the small arrow next to any procedure section to collapse or expand the contents of the procedure.




### Auto-completion

You can use the CTRL+SPACE keystroke combination, or type or pick items, to choose applicable items in the process editor.

- If you use CTRL+SPACE at the beginning of an empty line in the process editor, you'll see a list of categories:




  Click any of the categories to see a list of specific of items, then click an individual item to insert it into the editor.

- When you begin typing a function name, a list presents all functions that start with the characters you've typed. You can then click a function to insert it into the editor.
- When the cursor is positioned immediately before a parameter placeholder in a function, use CTRL+SPACE to display a list of valid parameter values, or in the case when there is only a single valid parameter value, to insert the value. If the cursor is positioned after a comma following a parameter, CTRL+SPACE lets you pick another parameter value of the same type as the previous parameter. For example, if the cursor is positioned after comma following a dimension name in the CubeCreate function, you can pick another dimension name to insert into the function.

## Code snippets

Click the snippet icon on the Script tab of the process editor to insert a predefined code snippet into the process.

The code snippets are provided for repetitive tasks that users perform often, making the process less error-prone. For more information, see "Insert a code snippet into a process" on page 810.

## Function list

You can click on the Script tab of the process editor to quickly insert a TurboIntegrator function into the process.

When you click , you see a list of function categories. Click a function category to see the available functions within the category, then click a function to insert it at the cursor in the process editor.

The function is inserted with placeholders for required parameters. You must replace the placeholders with valid parameter values. For example, if you use the function list to insert the ATTRN function into your process, you'll see this:

```
ATTRN( «dimension», «element», «attribute» );
```

You must replace the <<dimension>>, <<element>>, and <<attribute>> placeholders to successfully return a numeric attribute value with ATTRN.

## Shortcut keys

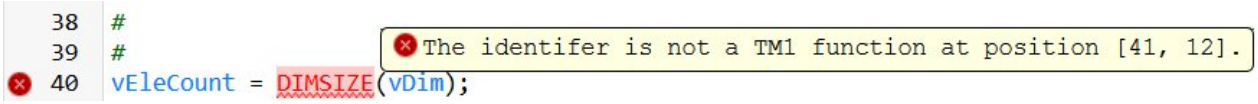
There are many shortcut keys available in the process editor. You can use these keys to edit statements, find and replace text, and navigate through the statements in the process editor.

To see a full listing of shortcut key combinations, click on the Script tab of the process editor.

## Error highlighting

Syntax errors in your process script are indicated by an error symbol next to the line number in which the error occurs. Syntax errors are also highlighted with a pink background in the process editor.

You can hover over either the error symbol or the highlighted syntax error to view a description of the error.




In this example, the DIMSIZ function is incorrectly spelled DIMSIZE. The incorrectly spelled function is highlighted and the error description indicates that the highlighted item is not a valid TM1 function.

## Line wrapping

To enable or disable line wrapping in the Script tab on the process editor in the editor, click .

### Text customization

To change the font and size of the text displayed in the Script tab on the process editor, click Aa.

### Locate the process in the Databases tree

You can click the **Find in tree** button to immediately locate the process in the Databases tree.

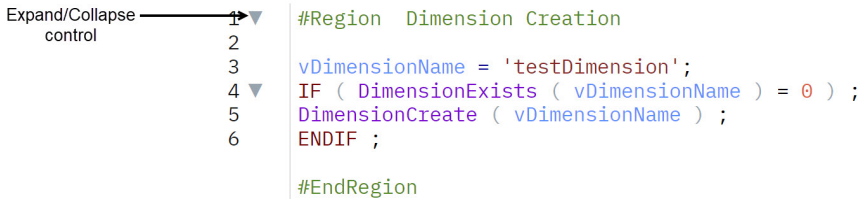



### *Using regions to define blocks of code*

A region defines a block of code with a user-defined name.

### About this task

When a region is included in your process script, you can click the accompanying Expand/Collapse control to expand or collapse the region. This is useful when working with long or complex processes.




### Procedure

1. On the **Script** tab of a process, enter `#Region`, followed by a space and a name for the region.
2. On subsequent lines, enter the code that you want to include in the region.
3. On a new line after the last line of code, enter `#EndRegion`.

   For example:

```
#Region  Dimension Creation

vDimensionName = 'testDimension';
IF ( DimensionExists ( vDimensionName ) = 0 ) ;
DimensionCreate ( vDimensionName ) ;
ENDIF ;

#EndRegion
```

## Run a process

You can run TurboIntegrator processes directly from the Databases tree or from within the process editor. Regardless of where you run the process, you cannot run a process that is being edited and has any unsaved changes. You must save (or discard) changes before you run the process.

### Procedure

1. To run a process from the Databases tree on a modeling workbench, expand the **Processes** group under the database that contains the process.
   a) Right-click the process.
   b) Click **Run Process**.
2. To run the current process in the process editor on a modeling workbench, click **Run**.
   You can click **Cancel** to cancel a running process in the process editor.

## View process error logs

If a TurboIntegrator process execution results in errors, you can view the error logs directly in Planning Analytics Workspace.

When a process results in errors, a new window opens indicating if the process completed with errors or aborted due to errors. The window also contains the error log generated by the process. The error log window is visible only to administrators and is displayed only when a process is executed directly from the process editor. The error log window does not display when errors are encountered when you run a process from a button.

If a process uses the ExecuteProcess or RunProcess function to execute one or more sub-processes, and there are errors in multiple processes, the window shows each process name with an arrow that you can click to expand or contract the error log for the individual process.

You can view or download stored edit logs at any time.

1. Right-click the **Processes** node for the database where you want to view logs.
2. Select the process log to view in the modal window or a separate tab.
   - To view the process log in the modal window, click **Error logs**.
   - To view the process log in a separate process log tab, click **Process logs**.
3. On the **Process error logs** page, select the log you want to review.
4. Click **View** to view the log or click **Download** to save the error log to the downloads directory of your browser.

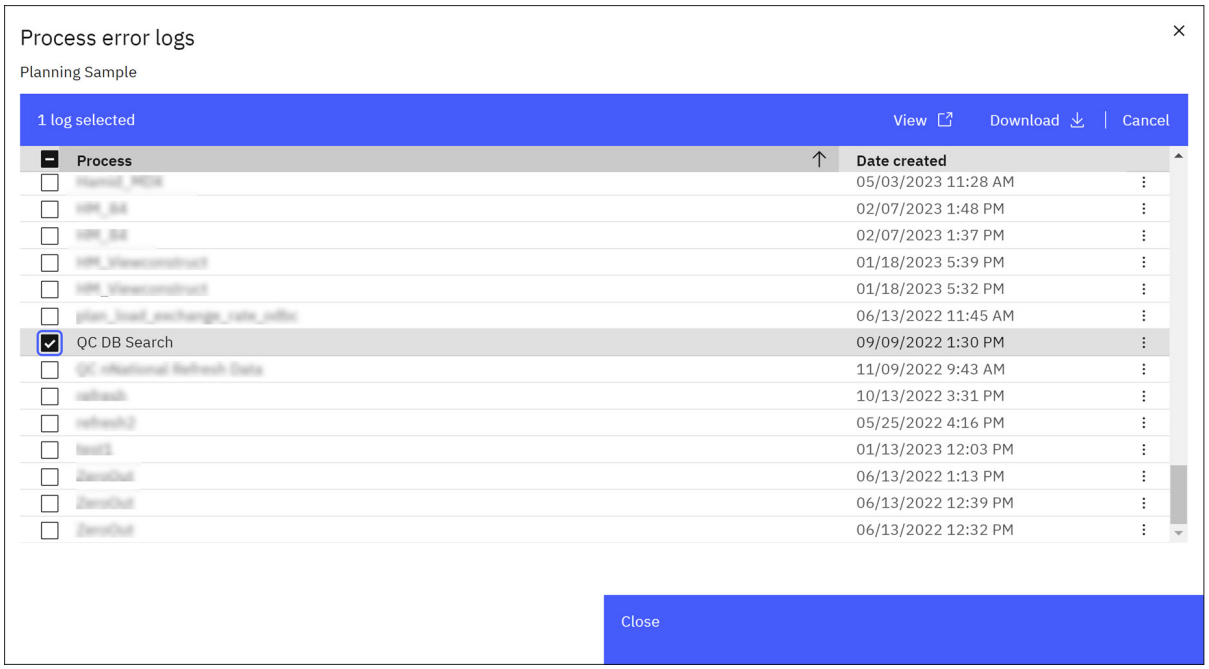
## Access process error logs from the Databases tree

You can view or download process error logs from the **Processes** node on the **Databases** tree in a modeling workbench.

### Procedure

1. On a modeling workbench, expand the database where you want to examine process log files.
2. Right-click the **Processes** node, then click **Error logs**.

   The **Process error logs** page displays all available logs, identified by **Process** name and **Date created**. You can click a column heading to sort the list of logs.



3. Select the individual log that you want to examine.
4. Click **View** to view the log in a temporary viewer or click **Download** to download the error log as a text file.

## Edit chores that involve a specific process

A process can be included in multiple chores. You can review a list of chores that include a process and edit selected chores.

### Procedure

1. Right-click the process on the **Databases** tree on a modeling workbench.
2. Click **Edit related chores**. If the process is not included in any chores, you receive notification and can cancel the procedure. Otherwise, continue to the next step.
3. On the **Edit chores** window, select the chores you want to edit.
4. Click **Edit**.

   Each of the chores you selected opens for editing on a separate tab on the modeling workbench.

## Remove a process from multiple chores

A process can be included in multiple chores. You can remove a process from multiple chores simultaneously.

### Procedure

1. Right-click the process on the **Databases** tree on a modeling workbench.
2. Click **Remove from chores**. If the process is not included in any chores, you receive notification and can cancel the procedure. Otherwise, continue to the next step.
3. On the **Remove chores** window, select the chores from which you want to remove the process.
4. Click **Remove**.
5. Click **Delete** when prompted for confirmation of removal.

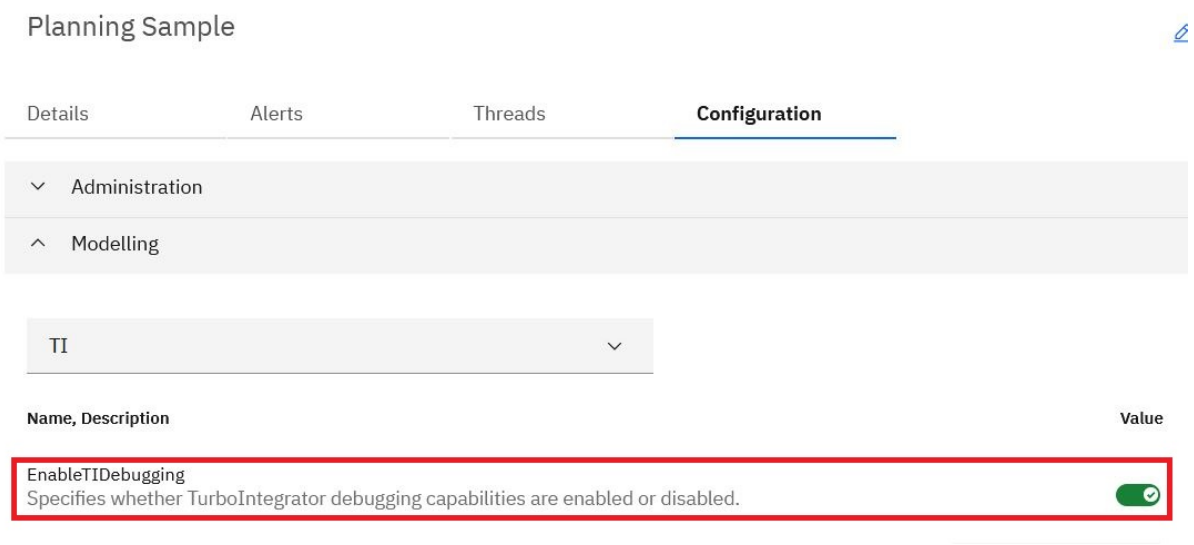
## Debug a process

Use the process debugger to set line breakpoints, continue to the next breakpoint, step over to the next line, and set conditional breakpoints.

Using breakpoints, you can stop the execution of code at a particular line. Conditional breakpoints are set by entering expressions. The expression must evaluate to true or false and can contain references to any variables and parameters, and use TM1 TurboIntegrator process functions. Potential expression errors can be corrected at run time.

### Before you begin

The configuration parameter **EnableTIDebugging** on the database must be set to true. By default, this parameter is set to false.




For more information, see Set database configuration parameters.

## About this task

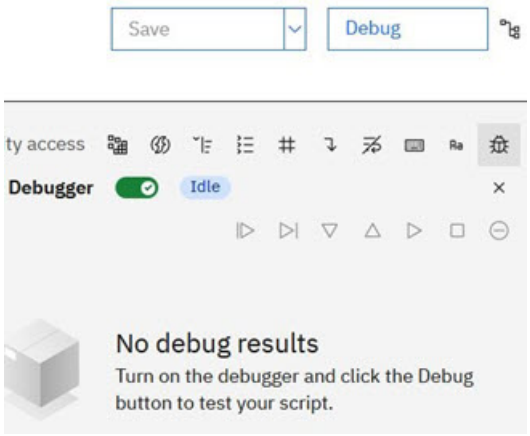
You can enable the debug mode only on one process at a time.

## Procedure

1. Open the process in the process editor.
2. Click the **Script** tab.
3. In the toolbar, click the debugger icon .

   The debugger panel is opened.
4. In the **Debugger** panel, switch the **Debugger** toggle to enable the debug mode.

   The **Run** button changes to **Debug** button.




5. Click the **Debug** button to start debugging the process.

   The debugger stops on the first executable line. The following buttons are now enabled, and you can use them to manage the debugging:

   - **Continue** - Continue processing to the next break point.
   - **Step over** - Stops at the beginning of the next line.
   - **Step in** - Goes to the next subprocess within the current ExecuteProcess function.

- **Step out** - Goes to the first line after the current ExecuteProcess function.
- **Continue to end** - Continue executing the process to completion.
- **Stop execution** - Cancels the process execution.
- **Remove all breakpoints** - Removes all debugging breakpoints.

Whenever the debugger pauses, the Variables section shows the current process variables and their values. These variables are read-only. You can also hover over any variable to see its current value.

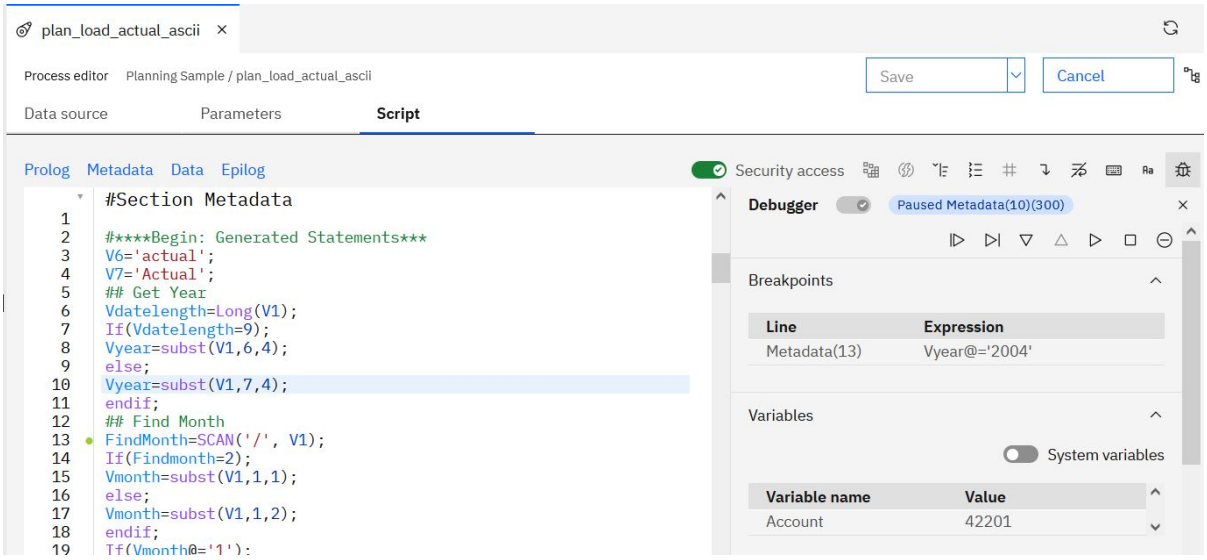
The Locks section displays information about the locks that are active when the debugger pauses.




6. To set a breakpoint, click the line number in the code editor.

   The **Breakpoints** section appears in the debugger panel. In this section, you can see the line of code for which breakpoints are set up, and the **Expression** field where you can add expressions for the breakpoints.
7. To set a conditional breakpoint, click the **Expression** field for the selected breakpoint, and type the expression.

   In the following example, the expression `Vyear@='2004'` is entered for the line 13 breakpoint to stop debugging when the input data is for year 2004.




   **Tip:** The breakpoint icon (the dot symbol) color changes to green.
8. If you want to, you can use a filter to specify specific variables that you want to examine during debugging.

The Variables section of the Debugger includes a **Filter variables** option. Click **Filter variables** and select a variable that you want to include in the debugging process. You can specify multiple variables for inclusion in debugging, but you must click **Filter variables** for each variable that you want to include.

If you do not set a filter to include at least one variable for inclusion during debugging, *all* variables are examined during debugging.

By default, only local process variables are shown when you click **Filter variables**. To include system variables for selection, toggle **System variables** to the **On** position.

9. Ensure that debugging is finished on the selected process before running it on another process.

### Debug a sub-process

The process debugger supports debugging a sub-process of a parent process.

#### About this task

Sub-processes are executed with the ExecuteProcess function in the parent process. There are two ways to debug a sub-process.

#### Procedure

- Place a breakpoint at the line with the ExecuteProcess function in the parent process, then click **Step in** when you reach the line during debugging.

  OR

- Place a breakpoint at the beginning of the sub-process, then start debugging the parent process. The debugger automatically switches to the sub-process and stops at the break point.

## Delete a process

You can delete a process directly from the Databases tree on a modeling workbench.

### About this task

You cannot delete a process that is included in any chores. You must remove the process from all chores before you can successfully delete the process.

### Procedure

1. Right-click the process in the Databases tree on a modeling workbench.
2. Click **Delete process**.
3. Click **OK** to confirm the deletion.

## Zero suppression in a TurboIntegrator process affects member ordering in a view

When you turn on zero suppression in a TurboIntegrator process, data in a view is ordered by when members (elements) were added to the dimension, instead of by the default dimension order (alphabetical) or by the defined subset order.

When a TurboIntegrator process is created with the parameter `pSuppressZero = 0` (suppress zero is turned off), it shows data for the dimension in descending alphabetical order or by the defined subset order, as expected. When a TurboIntegrator process is created with the parameter `pSuppressZero = 1` (suppress zero is turned on), it shows data for the dimension in the order that the members were added to the dimension.

This is an as-designed limitation based on the different types of cube data iteration associated with the suppress zeroes property on the view.

When zero suppression is off, TM1 performs an "expanse iteration" of the base cube data that visits all possible locations implied by the metadata. This type of iteration can be prohibitively slow.

When zero suppression is on, TM1 instead performs a "population iteration" that is driven by where actual data resides. This type of iteration can be much faster. However, the expanse iteration works by using member sorting indexes, while the population iteration uses member data coordinate indexes (of necessity, since that is how the data is stored.) When a new member is added, the sorting indexes of the members are reassigned to implement alphabetical ordering based on the member name. But the new member must also be assigned a new data coordinate index. This index will be above the existing range of data indexes that are already established at that time. This is the main reason for the difference in behavior.

This problem can be resolved by performing a TM1 database restart. After the database restart, the dimension will be freshly loaded into memory, at which time the sorting indexes and data coordinate indexes will match.

## Manage process attributes

Process attributes provide information about the purpose of processes.

### About this task

You can view and edit the values for process attributes, but you cannot create or delete attributes in the editor. Attributes for processes are created in a TurboIntegrator process that uses the process attribute functions.

### Procedure

1. On the **Database** tree on a modeling workbench, expand a database and right-click the **Processes** node.
2. Click **Process attributes**.

   A modified view of the }ProcessAttributes control cube opens, with individual processes on the row axis and attributes on the column axis.
3. Click a cell at the intersection of a process and an attribute, then enter or modify the attribute value.

   You can use cut, copy, and paste to manage attribute values. You can also use the standard view options to manage sets and to select multiple cells, rows, or columns in the editor.

# TurboIntegrator chores

You can create a chore to execute one or more TurboIntegrator processes in a specified sequence and at defined intervals. Chores are created and maintained on the modeling workbench.

A chore is composed of:

- A list of processes to be executed
- Optionally, runtime parameter values to be passed into processes upon execution from within the chore
- A start date and time for the initial execution of the chore
- An interval at which the chore is subsequently executed

Even when a chore has an execution interval defined, it can be manually executed at any time.

Once defined, chores can be activated and deactivated as required.

Access to chores functionality is controlled by your Planning Analytics Workspace role and user group security privileges. You must be a Planning Analytics Administrator or Modeler to be able to see chores in the Databases tree on a modeling workbench and have access to the chores shortcut menu options.

To create or edit chores, you must also be a member of the Admin or DataAdmin group on the Planning Analytics database.

# Create and edit chores

You can create chores to automate the execution of TurboIntegrator processes at defined intervals.

## About this task

You must be a Planning Analytics Administrator or Modeler to be able to see chores in the Databases tree and have access to the chores shortcut menu options. To create or edit chores, you must also be a member of the Admin or DataAdmin group on the Planning Analytics database.

**Note:** To use chores, you must be running a Planning Analytics database version 11.3 or later. Planning Analytics database 11.3 is the version that is included with Planning Analytics version 2.0.5. If you are using a Planning Analytics database version prior to 11.3, chores will not be available in the Databases tree and you cannot create or edit chores in Planning Analytics Workspace.

## Procedure

1. To create a new chore:
   a) On a modeling workbench, expand the database in the Databases tree on which you want to create the new chore.
   b) Right-click the **Chores** group.
   c) Click **Create chore**.
   d) You can change the database where the chore is stored by selecting a database from the list. The current database is selected by default.
   e) Enter a **Chore name**.
   f) Click **Create**.
2. To edit an existing chore:
   a) On a modeling workbench, right-click the chore in the Databases tree.
   b) Click **Edit chore**.
3. Specify which processes are included in the chore.
   a) Click **Add process**.
   b) You can add both user-defined processes and control processes to a chore. To narrow the focus of available processes, click either **Processes** or **Control processes**.
   c) In the list of **Available processes**, click the **Add** icon for each process you want to add to the chore. The processed you select are added to the list of **Selected processes**.
   d) To delete a process from the list of **Selected processes**, click the **Delete** icon for the process in the list.
   e) Click **Add**.
4. Set the order in which processes are executed.
   a) Click a process in the list of chore processes.
   b) Click ↑ or ↓ to move the process to the desired order in the chore. Repeat for any other processes until the desired execution order is set.
5. You must set parameter values for any processes in the chore that include parameters. The values you set are passed into the process upon execution within the chore.

   If any process includes parameters that have not been set, an Info icon warns you of this condition.




a) Click the parameter listing for the process.

b) Enter a value for each parameter.

c) Click **Save**.

6. Set the **Commit each process individually** option.

   When this option is enabled, each process in the chore is committed individually, as a series of separate transactions. Any locks that are required for a given process are held only for the duration of the process in which the lock is acquired.

   When this option is disabled, all processes in the chore are processed together and committed as a single transaction. Any locks acquired by any processes in the chore are held until the last process is complete. This means locks can potentially be held for very long periods of time.

7. Specify the initial chore execution date and time, and set a subsequent execution interval. Chore execution time is set using Coordinated Universal Time (UTC).

   a) Click the **Schedule** tab.

   b) Click the **Date** box and pick the initial execution data.

   c) Set the execution time, using your local time, in the **Time** box. The time you enter is converted and saved as UTC.

      Cloud hosts on IBM Planning Analytics on Cloud are configured for UTC time, which does not follow Daylight Saving Time. Even when the **Daylight Savings Time Sensitive** feature is enabled, the chore runs at the scheduled UTC time. Chores should be updated twice annually to ensure accurate execution time.

   d) Set a **Frequency** for chore execution, either **Daily** or **Custom**. If you select **Custom**, you can set the frequency of chore execution down to the second.

   e) Select the **DST Sensitive** checkbox if you want the chore time to be automatically updated when the daylight saving time starts or ends for the region in which the TM1 server is located.

8. Enable the **Scheduled** option to activate the chore schedule.

   If you do not enable this option, the chore is saved on your database, but is not executed until you enable the schedule or execute the chore on demand.

9. Click **Save**.

   You can optionally click **Run** to execute the chore immediately.

## Execute a chore on demand

You can execute a chore on demand at any time, regardless of the chore schedule.

### Procedure

1. On a modeling workbench, right-click the chore in the **Databases** tree.
2. Click **Run chore**.

   A status message in the Data tree region lets you know if the chore runs successfully or if errors prevent the chore from running.

## Enable or disable a chore schedule

You can enable or disable the schedule in the chore editor when creating or editing a chore. You can also enable or disable a chore schedule from the **Databases** tree.

### Procedure

1. On a modeling workbench, right-click the chore in the **Databases** tree.
2. Click the appropriate enable/disable schedule option.

   The options available vary depending on the current state of the chore.

   - If the chore schedule is currently enabled, the **Disable chore schedule** option is available.
   - If the chore schedule is currently disabled, the **Enable chore schedule** option is available.
   - If the chore has been created and saved, but is not yet complete (does not include any processes or does not have an execution frequency set), you won't see an enable/disable schedule option. You must complete the chore definition before you can enable or disable the schedule.

## Delete a chore

You can delete a chore directly from the **Databases** tree.

### Procedure

1. On a modeling workbench, right-click the chore in the **Databases** tree.
2. Click **Delete chore**.
3. Click **Delete** to confirm the deletion.

## Manage chore attributes

Chore attributes provide information about the purpose and use of chores.

### About this task

You can view and edit the values for chore attributes, but you cannot create or delete attributes in the editor. Attributes for chores are created in a TurboIntegrator process that uses the chores attribute functions.

### Procedure

1. On the **Database** tree on a modeling workbench, expand a database and right-click the **Chores** node.
2. Click **Chore attributes**.

   A modified view of the }ChoreAttributes control cube opens, with individual chores on the row axis and attributes on the column axis.
3. Click a cell at the intersection of a chore and an attribute, then enter or modify the attribute value.

   You can use cut, copy, and paste to manage attribute values. You can also use the standard view options to manage sets and to select multiple cells, rows, or columns in the editor.

## Running a chore at database startup

You can designate a chore as a "startup" chore that is processed when the database starts up.

To indicate that a chore should be run when the database starts up, use the configuration parameter StartupChores to identify a list of chores to be run before the server starts up.

Startup chores can be used as a way to set up the database before processing. Startup chores run before users login and before other chores begin processing.

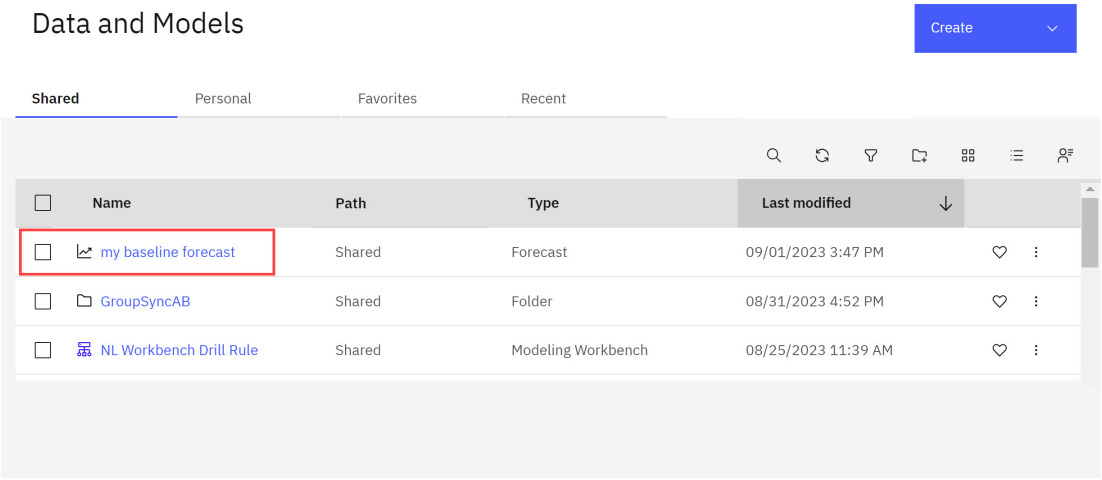
Since Startup chores are run before logins are allowed, the user can not monitor the Startup chores and therefore there is no way to cancel a Startup chore with the exception of killing the database process.

# Baseline forecasts

A baseline forecast is a Planning Analytics Workspace asset that administrators and modelers can create and share with other users in your organization.

A baseline forecast is unique from an ad-hoc forecast that you can perform on a view. A baseline forecast is saved as an asset that is available to other Planning Analytics Workspace users on both the **Data and Models** page and the **Reports and Analysis** page. Baseline forecasts also allow forecasting on up to 8,000 rows, which is much more than an ad-hoc forecast can accommodate. You can use an external scheduler to schedule baseline forecasts.

Administrators, modelers, and analysts with the appropriate permissions can run a baseline forecast at any time and use the forecast to improve the planning process. Consumers cannot run baseline forecasts.




There are several steps to creating a baseline forecast:

## Create the baseline forecast asset and set forecast details

When you initially create a baseline forecast, you must specify the Planning Analytics database and cube that provide the data from which the forecast is generated.

### About this task

You must be an administrator or modeler to create a baseline forecast.

### Procedure

1. Click the Planning Analytics Workspace home menu, then click **New** > **Baseline forecast**.

   You can also create a baseline forecast from the Data and Models page. Click **Create** > **Baseline forecast**.
2. Select the **Database** that contains the cube from which you want to generate a forecast.
3. Select the **Cube** that contains the data you want to use as the basis of your forecast.
4. Specify whether you want the baseline forecast to be a univariate or multivariate forecast.
   - To specify a multivariate forecast, click the **Univariate** or **Multivariate** icon until it is green with the **Multivariate** label.
   - To specify a univariate forecast, click the **Univariate** or **Multivariate** icon until it is grey with the **Univariate** label.

5. Optionally, enter a **Description** of the forecast to help users understand the purpose of the forecast. The description is visible to other users when they open the baseline forecast asset from the Data and Models page..
6. Click **Next**.

### What to do next

"Define a baseline forecast timeframe" on page 833

## Define a baseline forecast timeframe

You must identify one or more dimensions that represents time in the cube you use for forecasting.

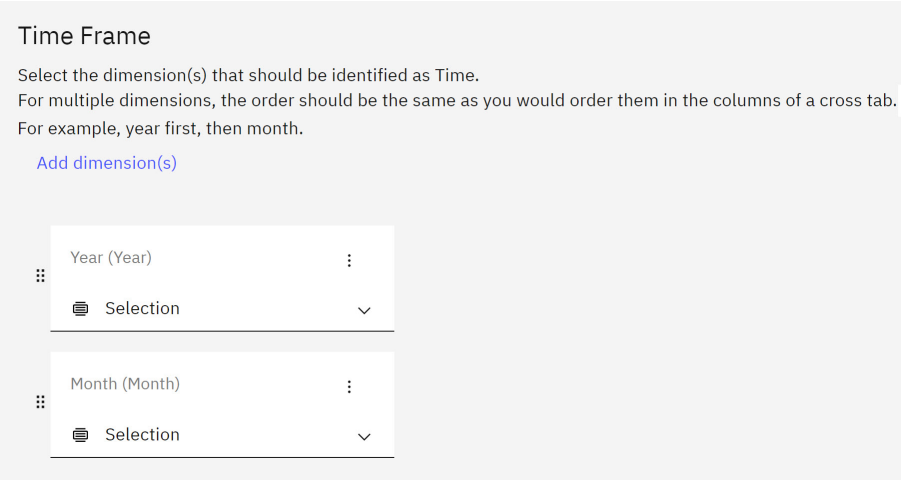
### About this task

The time dimension (or dimensions) should include members for future time periods that correspond to the timeframe for which you want to create a forecast.

### Procedure

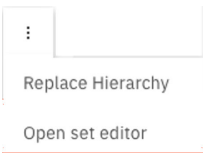
1. On the **Timeframe** page, click **Add dimension(s)**.
2. Select the dimension or multiple dimensions that represent time in your cube.

   In the case of multiple dimensions, you should order the dimensions as you would in an exploration, with the more general time period followed by the more specific time period. For example, if your cube includes the dimensions Year and Month, your configuration would look like this.




   You can click a dimension and drag it to a different position if necessary.
3. When you specify a time dimension, the dimension uses the default hierarchy and the set is `Selection`.

   Click the **Options** icon to change the hierarchy for a time dimension or to define a new set.




   If you define a new set, you must explicitly save the set with a name.

   Click ⌄ to select a different set or level for the time dimension.
4. Click **Select start period** to define the start of your forecast.
5. The **Forecast period end** is automatically set to the last member of your time dimension (or dimensions, in the case of multiple time dimensions). You can click the menu icon ⌄ to pick a different end period.

6. Optionally, you can ignore some historical time periods in your forecast.

   To ignore periods after a specific point in time, click **After**, then click **Select ignore time periods** and select the start of the historical time period that should be ignored. Any period after the selected period is ignored when generating a forecast.

   To ignore an individual time period, click **Individual**, then click **Select ignore time periods** and select the individual historical time period that should be ignored
7. Click **Next**.

### What to do next

"Specify the scope of a baseline forecast" on page 834

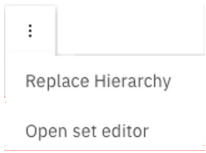
## Specify the scope of a baseline forecast

The scope of the forecast defines the members for which you want to generate forecast values.

### Procedure

1. Drag and drop the dimensions that contain the members for which you want to generate a forecast from the **Context** area to the **Forecast members** area. Think of these members as the members on the rows in an exploration. You can specify up to three dimensions in the **Forecast members** area.

   **Important:** By default, only the first member in a dimension is added to the forecast scope. To change the forecast member or to add additional members to the scope, complete Step 2.
2. Click the **Options** icon to change the hierarchy for a forecast member dimension or to define a new set.

   If you define a new set, you must explicitly save the set with a name. The set will then be available for selection when you click ⌄ to select a different set or level for a forecast member dimension.
3. As with forecast member dimensions, you can click the **Options** icon to modify the hierarchy or set for a context member dimension. You can also click ⌄ to select a different set or level for dimensions in the **Context members** area.
4. If you want to let users override context member or forecast member dimensions and specify a custom scope when they run the baseline forecast, enable the **Allow override** option for dimensions in the **Context members** area or the **Forecast members** area.
5. Click **Next**.

### What to do next

If you defined a mutivariate baseline forecast, specify the variables of a multivariate baseline forecast. Otherwise, define advanced baseline forecast settings.

## Specify the variables of a multivariate baseline forecast

If you are creating a multivariate forecast, specify the variables for the baseline forecast.

Perform the following actions on the **Multivariate** page.

**Add variables**
: You can add up to two variables for the forecast. These variables can be for the same cube or for a different cube.

**Edit variable**
: Edit any existing variables that you already added.

**Delete variable**
Remove existing variables from the baseline forecast.

After you define your variables, click **Next** to define advanced baseline forecast settings.

## Define advanced baseline forecast settings

Advanced settings are optional when you create a baseline forecast. If you choose to specify advanced settings, you can use the **Advanced settings** page to set seasonality, set the scope of historical data, define a confidence interval, define where you want the forecast values to appear, and set data spreading.

The following options are available on the **Advanced settings** page.

**Seasonality**
The seasonality with which to build the model. Seasonality is when the specified timeframe has a predictable cyclic variation. For example, during a holiday period each year.

**Auto-detect** is enabled by default. **Auto-detect** automatically detects seasonality by building multiple models with different seasonal periods and choosing the best one.

If you turn **Auto-detect** off, you can specify seasonality by entering a non-negative integer, such as 0, 1, 2, 3 in the **Enter seasonality interval** field.

To specify a non-seasonal model, set the **Enter seasonality interval** to 0 or 1. A model with user specified seasonality is displayed only if the seasonal model is more accurate than all of the non-seasonal models.

**Confidence level**
The certainty with which the true value is expected to be within the specified range.

You can select three different confidence intervals: 90%, 95%, and 99%. The default is 95% and the lower and higher bound define the range at which you can be 95% confident that the true value lies within that range.

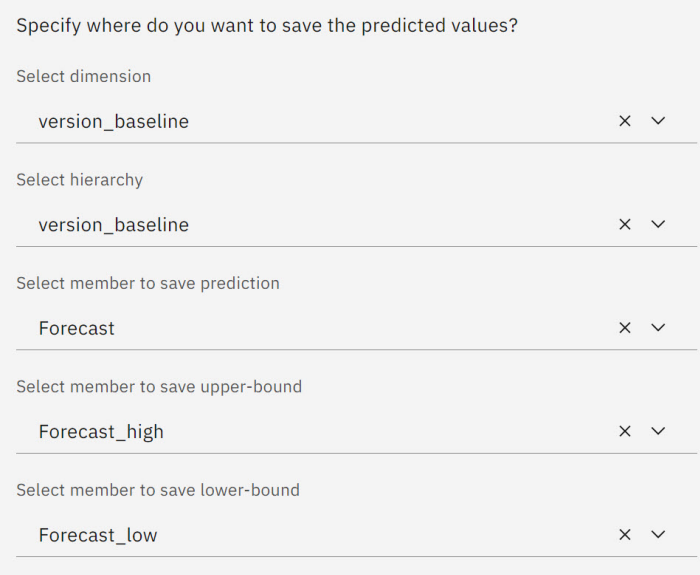
**Adjust outliers**
When this option is turned on, all detected outliers are corrected before performing the forecast.

**Spread forecast values**
Define the data spreading method you want to use to spread data to consolidated cells.

You can use the Proportional and Relative proportional data spreading methods. For more information, see “Apply data spreading to a forecast” on page 538.

**Where do you want to save the predicted values**
This section lets you identify the particular dimension, hierarchy, and member where you would like to save your forecast values, as well as where you would like to save upper bound and lower bound values.

For example, you would typically save forecast values to the Forecast member of a Version dimension.




After you define your advanced settings, click **Next** to view the baseline forecast summary and activate the forecast.

# View the baseline forecast summary and activate the baseline

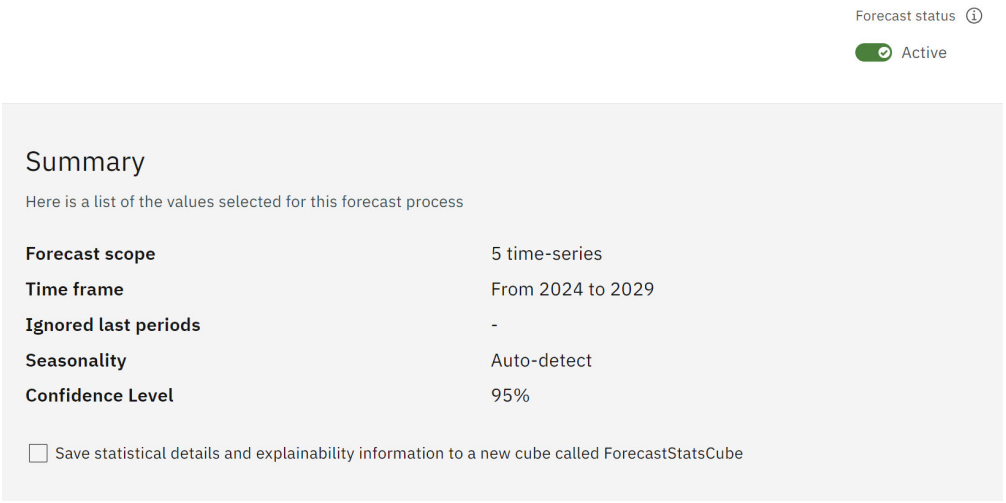
The Summary page lets you review the parameters of your forecast and activate the forecast so other use can run it.

## About this task

In the top right corner of the dialog, you will see the 'Forecast status' which indicates whether the forecast is active or inactive. If the status is set to 'Inactive', neither analysts nor consumers will be able to open the forecast. None of the users will be able to run the forecast as well. However, if the status is set to 'Active', all users with access to the database and cube will be able to run the forecast.

## Procedure

1. Review the summary of your baseline forecast. If anything looks amiss, you can click **Back** to return to previous configuration pages.




2. Optionally, you can save the statistical details and other information about the forecast.

If you enable the **Save statistical details ...** option, the statistical details that are generated when you run a forecast are saved in the `ForecastStatsCube`, where you can analyze the details after a forecast is complete. If this option is not enabled, the statistical details are not saved anywhere and cannot be reviewed. If you also selected the **Allow override** option, you can also allow your users choose the slice to store the statistical details.

The first time you enable the **Save statistical details ...** option on a Planning Analytics database you are asked to create the `ForecastStatsCube`.

The `ForecastStatsCube` saves statistical data about baseline forecast executions. It contains three dimensions:

**ForecastResultRun**
: Initially, this dimension contains members numbered 1 through 5, representing individual baseline forecasts. When you select a slice to which you want to save statistical details, you specify one of the members in this dimension to save details data. After you initially create the `ForecastStatsCube`, you can modify this dimension to contain other members to store statistical details for other baseline forecasts.

**ForecastResultTimeseries**
: This dimension has members corresponding to time series as members.

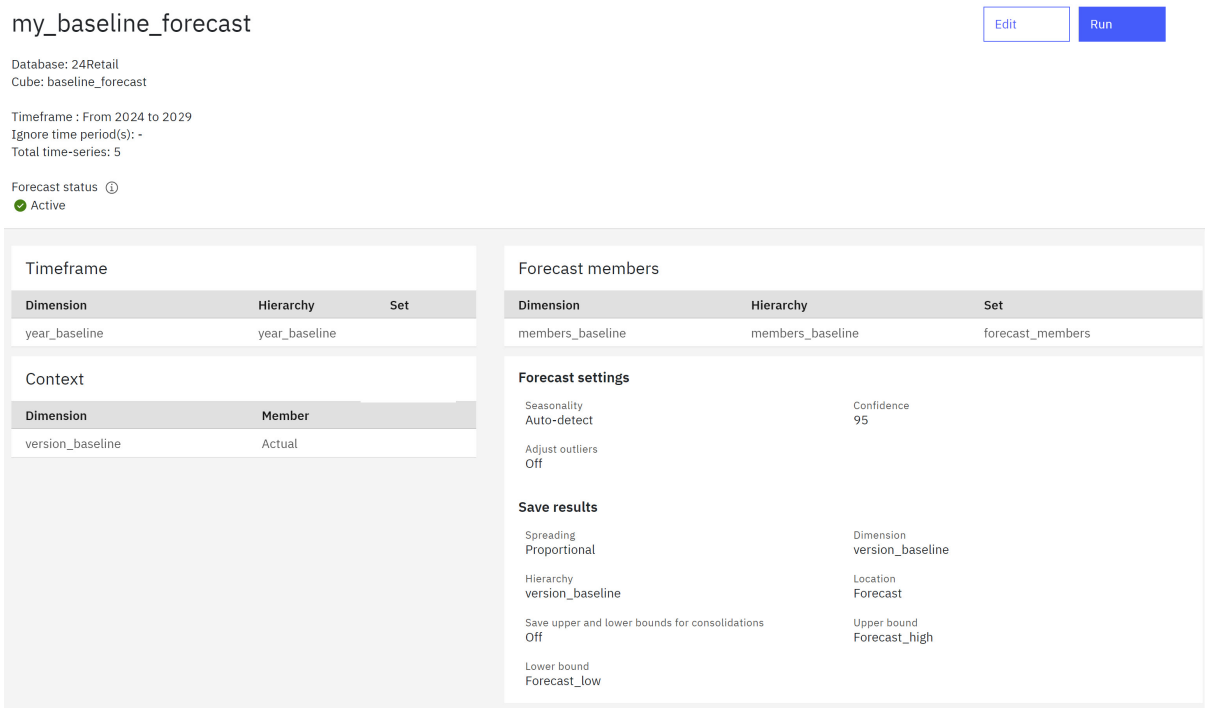
**ForecastResultOutput**
: This dimension has all the statistical details categories as its members.

a) Click **Create**.

b) Use the **Select a slice** option to pick the member of the `ForecastResultRun` dimension to receive the statistical data.

   If you selected the **Allow override** option for the **Save statistical details ...** option, your users can choose a different slice to store the statistical details when they run the baseline forecast.

3. Toggle the **Forecast status** to **Active** to enable running the forecast and to make the forecast available to other users.
4. Click **View baseline** to open the baseline forecast page, which provides an overview of your forecast. You can edit or run the baseline forecast from this page.



### What to do next

"Run a baseline forecast" on page 838

## Edit a baseline forecast

Administrators and modelers can edit baseline forecasts.

### Procedure

1. Click the **Data and Models** tile on the Planning Analytics Workspace home page.
2. Click the baseline forecast in the list of assets on the **Data and Models** page.
3. Click **Edit** on the baseline forecast page.

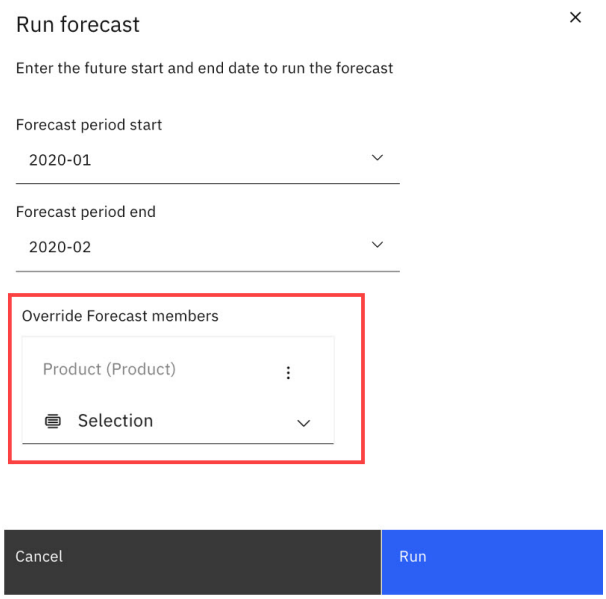
## Run a baseline forecast

Administrators, modelers, and analysts with the appropriate permissions can run a baseline forecast at any time. Consumers cannot run baseline forecasts.

### Procedure

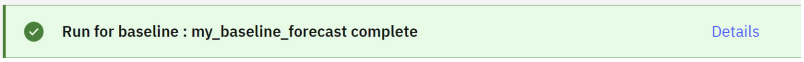
1. Click the **Data and Models** tile on the Planning Analytics Workspace home page.
2. Click the baseline forecast in the list of assets on the **Data and Models** page.
3. Click **Run** on the baseline forecast page.
4. The **Forecast period start** and **Forecast period end** values are set using the periods that were specified when the baseline forecast was created, but you can modify either of these values. Click either period value and select from any of the members available in your timeframe dimension.
5. If the baseline forecast is configured to allow the scope of the forecast to be modified at runtime, you can use the **Override Forecast members** option to pick an existing set or level for a forecast dimension or open the set editor and define a new set.

   Click the ⌄ menu to select an existing set or level. Click the **Action** ⋮ icon to open the set editor.




6. If you enabled both the **Save statistical details ...** option and the **Allow override** option to allow a different slice to be chosen at runtime, you can select a slice from the **Select a slice ...** field.
7. Click **Run**.

   A status notification message appears on the baseline forecast page when the forecast finishes running.




If the forecast runs without errors, the status message is green. If there are error messages in one or more timeseries, the status message is yellow. If all the timeseries fail, the status message is red. You can click **Details** to view more information about the forecast execution.

If you enabled the **Save statistical details ...** option on the baseline forecast summary, you can view the `Error_Message` column on the `ForecastStatsCube` to review the full error message for every failed timeseries.

After the forecast is runs successfully, you can view the forecast values in the cube for which you created the baseline forecast.

# Schedule a baseline forecast

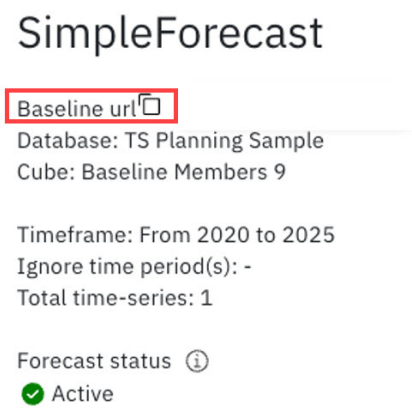
Scheduling a baseline forecast as a service is supported on Planning Analytics as a Service and most configurations of Planning Analytics Local. Scheduling a forecast is not supported in Planning Analytics on Cloud and Planning Analytics Local configured with Windows integrated authentication (mode 3).

## Retrieve the baseline forecast URL

You can use the baseline URL to schedule a baseline forecast with your external scheduler.

### Procedure

1. Click the **Data and Models** page on the Planning Analytics Workspace home page.
2. Click the baseline forecast for which you want to retrieve the URL.
3. On the forecast page, click the **Copy** icon next to **Baseline URL** to copy the URL to your clipboard.




## Schedule a baseline forecast on Planning Analytics Local

Use these commands when scheduling a baseline forecast on Planning Analytics Local.

### Procedure

1. To run the baseline endpoint: `curl --location --request POST 'http://<paw_local>/<baseline_url>' --header 'Authorization: Basic base64(<username>:<password>)'`

   When SSL is configured, pass the certificates with the `curl` command (recommended) or use the `-k` option to disable it. Once a baseline forecast is started, the status of the baseline asset changes to running. After the forecast is complete, it goes back to ready. While a forecast is running on a baseline asset, another forecast cannot be started unless the first one is complete and the asset returns to ready status.

2. To check the baseline status endpoint: `curl 'http:// <paw_local> /api/v0/predict/api/v1/BaselineStatus/<baselineID> ' --header 'Authorization: Basic base64(<username>:<password>)'`

   **Tip:** The `baselineID` can be found in the baseline URL as the value of the property named `id`.
3. If a baseline forecast gets stuck while running, you can reset the status back to ready: `curl --location --request POST 'http://<paw_local>/api/v0/predict/api/v1/ResetBaseline/<baselineID>' --header 'Authorization: Basic base64(<username>:<password>)'`
4. To override context members, statistical details cube slices, and sets, you must send a payload (request body) with the RunBaseline endpoint.

   To run the basement endpoint: `curl --location --request POST 'http://<paw_local>/<baseline_url>' --header 'Authorization: Basic base64(<username>:<password>)' -H "Content-Type: application/json" -d ‘json_payload_containing_sets_and_context’`

   The following is an example payload:

```
{
    "serverName": "Planning Sample",
    "statCubeSlice": " [ForecastResultRun].[ForecastResultRun].[3]”,
    "sets": [
        {
            "set": "all_leaves",
            "dimensionID": "Sample_members",
            "setIsOverridden": true,
            "hierarchyID": "Sample_members",
            "hierarchy": "Sample_members",
            "name": "Sample_members",
            "member": "all_leaves",
            "setType": "NAMED_SET",
            "mdx": "{{TM1FILTERBYLEVEL(TM1SUBSETALL([Sample_members].[Sample_members]) ,
0)}}"
        }
    ],
    "context": [
        {
            "dimensionID": "test_version",
            "hierarchyID": "test_version",
            "hierarchy": "test_version",
            "name": "test_version",
            "member": "Forecast",
            "contextIsOverridden": true,
            "setType": "UNNAMED_SET",
            "mdx": "{[test_version].[test_version].[Forecast]}"
        },
        {
            "dimensionID": "Months-Consolidated",
            "hierarchyID": "Months-Consolidated",
            "hierarchy": "Months-Consolidated",
            "name": "Months-Consolidated",
            "member": "1",
            "contextIsOverridden": true,
            "setType": "UNNAMED_SET",
            "mdx": "{[Months-Consolidated].[Months-Consolidated].[1]}"
        }
    ]
}
```

## Schedule a baseline forecast on Planning Analytics as a Service

Use these commands when scheduling a baseline forecast on Planning Analytics as a Service.

### Procedure

1. Create an IBM Cloud API key if you do not have one.

   If you already have a key, you can skip these steps. Copy your existing key, then proceed directly to step 2.

a) Log in to your Planning Analytics as a Service instance.

b) Click the user profile and click **Manage API Keys**.

c) Click **Generate key**.

d) Specify a name for the key, then click **Generate key**

e) Copy or download the API key.

2. To run the baseline endpoint:

   ```
   curl --location --request
   POST https://<paw_url>/api/<tenantID>/v0/predict/api/v1/RunBaseline?
   id=<baseline_id>&startDate=<start_date_tuple>&endDate=<end_date_tuple> --
   header 'Authorization: Basic base64(apikey:<key_generated_in_step1>)'
   ```

3. To check the baseline status endpoint:

   ```
   curl 'http:// <paw_local> /api/<tenantID> /v0/predict/api/v1/
   BaselineStatus/<baselineID>' --header 'Authorization: Basic
   base64(apikey:<key_generated_in_step3>)'
   ```

   **Tip:** The `baselineID` can be found in the baseline URL as the value of the property named `id`.

4. If a baseline forecast gets stuck while running, you can reset the status back to ready:

   ```
   curl --location --request POST 'http://<paw_local>/api/<tenantID> /v0/
   predict/api/v1/ResetBaseline/<baselineID>' --header 'Authorization: Basic
   base64(apikey:<key_generated_in_step1>)'
   ```

5. To override context members, statistical details cube slices, and sets, you must send a payload (request body) with the RunBaseline endpoint.

   To run the basement endpoint: `curl --location --request POST https://<paw_url>/api/<tenantID>/v0/predict/api/v1/RunBaseline?id=<baseline_id>&startDate=<start_date_tuple>&endDate=<end_date_tuple> --header 'Authorization: Basic base64(apikey:<key_generated_in_step3>)' -H "Content-Type: application/json" -d 'json_payload_containing_sets_and_context'`

   The following is an example payload:

```
{
    "serverName": "Planning Sample",
    "statCubeSlice": " [ForecastResultRun].[ForecastResultRun].[3]",
    "sets": [
        {
            "set": "all_leaves",
            "dimensionID": "Sample_members",
            "setIsOverridden": true,
            "hierarchyID": "Sample_members",
            "hierarchy": "Sample_members",
            "name": "Sample_members",
            "member": "all_leaves",
            "setType": "NAMED_SET",
            "mdx": "{{TM1FILTERBYLEVEL(TM1SUBSETALL([Sample_members].[Sample_members]) ,
0)}}"
        }
    ],
    "context": [
        {
            "dimensionID": "test_version",
            "hierarchyID": "test_version",
            "hierarchy": "test_version",
            "name": "test_version",
            "member": "Forecast",
            "contextIsOverridden": true,
            "setType": "UNNAMED_SET",
            "mdx": "{[test_version].[test_version].[Forecast]}"
        },
        {
            "dimensionID": "Months-Consolidated",
            "hierarchyID": "Months-Consolidated",
            "hierarchy": "Months-Consolidated",
            "name": "Months-Consolidated",
```

```
            "member": "1",
            "contextIsOverridden": true,
            "setType": "UNNAMED_SET",
            "mdx": "{[Months-Consolidated].[Months-Consolidated].[1]}"
        }
    ]
}
```

# Control objects

Control objects perform integral tasks in IBM Planning Analytics.

Most control objects are generated by the system and can be considered "system" control objects. System control dimensions, system control cubes, and system control processes are used to manage objects, maintain users and groups, apply and monitor security, monitor performance, and store object attributes and properties.

All control object names are prefixed with }, for example: `}Capabilities`. Control objects are available from the **Control Objects** node in the Databases tree. Most control objects are generated by the system. Only modelers and administrators can see or manage control objects.

To find out more about system control dimensions, see Control Dimensions.

To find out more about system control cubes, see Control Cubes.

Because the visibility of control objects is limited to modelers and administrators, there are some circumstances where non-system control objects are useful.

As an example, consider cubes used for lookups in MDX statements. The end user needs READ access to this sort of cube, but a modeler or administrator probably doesn't want to expose these cubes in the list of cubes available to most users. In this case, a control object allows the required access to all users while hiding the object from users who are not modelers or administrators.

A modeler or administrator can create non-system control dimensions, cubes, and processes.

## Create a control dimension

You must create control dimensions using the procedure described here. If you attempt to create a control dimension from the **Dimensions** node on a database, you receive a validation error.

### Procedure

1. Open a Modeling workbench.
2. In the **Databases** tree, expand the database where you want to create the control dimension.
3. Right-click the **Control Objects** node, then click **Create control dimension**.
4. Enter a **Control dimension name**.

   The leading } character is already in place.

   The name you enter is validated against a list of reserved system control object names and prefixes. If you enter a reserved name or prefix you get notification that the name is reserved and cannot be used.
5. Click **Create** to create the new control dimension.

   After you create the control dimension, you can add members, set attributes and weights, and create sets as you would for a regular dimension.

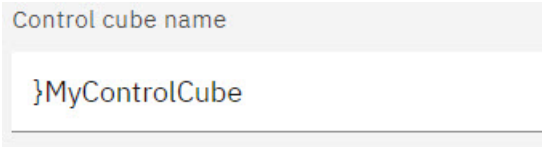
## Create a control cube

You must create control cubes using the procedure described here. If you attempt to create a control cube from the **Cubes** node on a database, you receive a validation error.

### Procedure

1. Open a Modeling workbench.
2. In the **Databases** tree, expand the database where you want to create the control cube.
3. Right-click the **Control Objects** node, then click **Create control cube**.
4. Enter a **Control cube name**.

   The leading } character is already in place.

   The name you enter is validated against a list of reserved system control object names and prefixes. If you enter a reserved name or prefix you get notification that the name is reserved and cannot be used.
5. Click **Next**.
6. Select the dimensions you want to include in the cube from the **Dimensions** list. When you select a dimension, it is added to the **Selected** list.
7. Optionally, you can change the order of dimensions in the cube by clicking a dimension handle and dragging it to a new position or by clicking the **Up** or **Down** arrow next to a dimension.
8. Click **Create** to create the new control cube.

   After you create the control cube, you can enter or import data, set attributes and properties, and manage the object as you would for a regular cube.

## Create a control process

You must create control processes using the procedure described here. If you attempt to create a control dimension from the **Processes** node on a database, you receive a validation error.

### Procedure

1. Open a Modeling workbench.
2. In the **Databases** tree, expand the database where you want to create the control process.
3. Right-click the **Control Objects** node, then click **Create control process**.
4. Enter a **Control process name**.

   The leading } character is already in place.

   The prefix `}Drill_` is reserved. You cannot create a control process name that starts with this prefix.
5. Click **Create** to create the new control process.

   After you create the control process, you can add a data source, set variables, define parameters, and create scripts as you would for a regular process.

# Configure drill-through to reveal detailed data

You can associate a cube cell with extra data, which can provide underlying detail for the cell or other information relevant to the cell. When you complete the configuration that is required to establish this association, users can drill through to the additional data directly from an Exploration view.

Drill-through configuration consists of two components:

- **Drill rule** - Defines the relationship between the cell and the detailed data.
- **Drill process** - Defines the detailed data that you want to associate with a cell.

When both a drill rule and a drill process are in place for a cell, you can right-click the cell and click **Drill through** to reveal underlying detail. If multiple drill processes are associated with a cell, you can pick the one you want to use.

For more information, see "Drill through to detailed data" on page 617.

# Create a drill rule

A drill rule is a unique type of rule that associates view cells with related data. The related data, which is opened through a TurboIntegrator process, can be a cube view or an ODBC source.

## About this task

Drill rules are distinct from cube business rules. While business rules define the data for specified cells in a cube, drill rules identify the relationship between specified cells and related (frequently more detailed) data.

There can be some interplay between business rules and drill rules. For example, you might have a Cube A that uses business rules to pull values from a separate Cube B. You might then create a drill rule (and an accompanying drill process) that lets users drill-through from Cube A to Cube B to review the source values in full context.

Similarly, if the values in a cube were imported from an ODBC source, you could create a drill rule and process that lets users drill-through from the cube to a grid-formatted view of the ODBC source.

A drill rule uses the syntax `['area_definition']=S:'drill_process_name';`. *area_definition* is the section of the cube in which you want to enable drill-through. *drill_process_name* is the name of the TurboIntegrator process that is executed when you execute a drill-through from a cell that is included in the *area_definition*. You can learn more about setting up drill processes in "Create a drill process" on page 846.

## Procedure

1. In edit mode, go to the database in the Data tree where you want to create a drill rule and expand the database to reveal the cubes.
2. Right-click the cube for which you want to create drill rules.
3. Click **Drill**, **Create drill rules**.

   The drill rules editor opens. This editor is identical to the business rules editor, but the icon on the upper left of the editor can help you distinguish between the two.

   - Drill rules editor icon 
   - Business rules editor icon 

   For each cube area you want to associate with detailed data, continue with steps 8 - 11.
4. Enter the area definition for the region of the cube where you want to enable drill-through.

   The area definition can be as broad as the entire cube or as specific as a single cell.

   To specify the entire cube, enter an empty bracket set `[ ]`.

   To specify a single member, enter the member name enclosed in single quotation marks within brackets. For example, `['Price']`.

   To specify an area identified by multiple members, enter the members within brackets. Enclose each member name in single quotes and use a comma to separate member names. For example, the area definition `['Price','France','Jan']` applies to any cell identified by the members Price, France, and Jan.

**Tip:** You can use the auto-complete or drag-and-drop features of the rules editor to simplify area definition.

5. After you enter the area definition, type `=S:` in the rules editor.
6. After the colon, enter the name of the drill process that will be run when a user initiates drill-through from a cell within the area definition. Enclose the drill process name in single quotation marks.

   **Important:** When you create a drill process, Planning Analytics creates a control process with the `}Drill_` prefix. For example, when you create a drill process named `PriceCubeToSource`, Planning Analytics creates the `}Drill_PriceCubeToSource` control process. Do not include the `}Drill_` prefix when you specify a drill process name in a drill rule.

   You can associate more than one drill process with an area. Enclose all drill processes, separated by commas, within a set of single quotation marks.
7. Enter a semi-colon at the end of the rule to indicate the conclusion of the drill rule statement.
8. Click  .

### Results

When a drill rule and associated drill processes are in place, you can right-click a cell that is included in the rule area definition and view underlying data. If the area is associated with a single drill process, the process is immediately run. The underlying data is displayed in either an exploration view for a cube view source, or a grid for an ODBC source.

If you initiate drill-through from a cell that is associated with multiple drill processes, you can choose which process you want to run.

#### Example

`[ ]=S:'PriceDrill';` this drill rule runs the `PriceDrill` drill process when you initiate drill-through from any cell in the cube.

`['Price']=S:'ODBCSourceDrill';` this drill rule runs the `ODBCSourceDrill` drill process when you initiate drill-through from any cell in the cube that is identified by the `Price` member.

`['Jan']=S:'PriceDrill, ODBCSourceDrill';` this drill rule lets you choose to run either the `PriceDrill` or `ODBCSourceDrill` drill process when you initiate drill-through from any cell in the cube that is identified by the `Jan` member.

`['Price','France','Jan']=S:'PriceDrill';` this drill rule runs the `PriceDrill` drill process when you initiate drill-through from any cell in the cube that is identified by the `Price`, `France`, and `Jan` members.

## Edit a drill rule

Planning Analytics Workspace modelers and administrators can edit drill rules.

### Procedure

1. In edit mode, go to the database in the Data tree where you want to edit a drill rule, and expand the database to reveal the cubes.
2. Right-click the cube for which you want to edit drill rules, and click **Drill**, **Edit drill rules**.

   Use the drill rules editor to modify your drill rules.

## Delete a drill rule

Planning Analytics Workspace modelers and administrators can delete drill rules.

### Procedure

1. Log in to Planning Analytics Workspace with a user that has the Modeler or Administrator role.
2. Make sure that you are in edit mode.
3. In the Data tree, navigate to the database where you want to delete a drill rule, and expand the **Cubes** group.
4. Right-click the cube for which you want to delete drill rules.
5. Click **Drill**, **Delete drill rules**, and click **OK**.

## Create a drill process

A drill process is a special type of TurboIntegrator process that is called from a drill rule to display underlying data associated with a view cell. A drill process is run when you initiate drill-through from an exploration view.

### About this task

A drill process can open a cube view or a DB connection (ODBC) source.

### Procedure

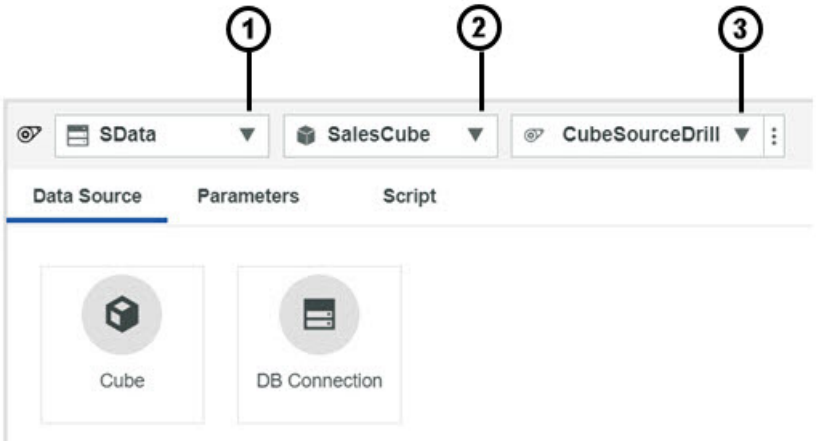
1. In edit mode, go to the database in the Data tree where you want to create a drill process, and expand the database to reveal the cubes.
2. Right-click the cube for which you want to create a drill process.

   This cube, from which a drill-through originates, is called the origination cube.
3. Click **Drill**, **Create drill process**.
4. Enter a name for the process, then click **Create**.

   **Tip:** For best practice, use a drill process name that identifies the origination cube for the drill process. For instance, if you create a process to drill from a cube that is named PriceCube to an ODBC source, you might name the drill process PriceCubeToODBCSource. This naming convention makes it easier to identify a drill process when you initiate drill-through from a cube view and must select from multiple drill processes that are associated with the cube.

   The drill process editor opens. This editor is nearly identical to the regular process editor, but has only two data source options: **Cube** and **DB Connection**. These data sources are the only ones that are supported for drill-through.

   Additionally, the drill process editor includes a cube selector, which is not part of the regular process editor. Drill processes are associated with a specific cube. If you pick a new cube from the cube selector, the process selector contains any drill processes associated with the selected cube.




**1**
Database selector.

**2**
Cube selector.

**3**
Drill process selector.

5. To define a drill process with a Cube source, complete these steps:
   a) Click **Cube**.
   b) Select the cube that contains the view that you want to open, then click **Next**.

      You can open any cube that resides on the same database as the current drill process.
   c) Select the view that you want to open with the drill process. If such a view does not exist, click **Create view** to define a new view.
   d) Click **Load Preview**.
   e) Click **Validate**.

      If validation fails, see Troubleshooting.
   f) Click **Save**.

      TM1 saves the drill process as a TurboIntegrator process, but prefixes the name you assigned in step 8 with the string }Drill_. For example, if you save a drill process with the name PriceCubeToODBCSource, TM1 saves the process as }Drill_PriceCubeToODBCSource. Drill processes appear under the Control Objects for your database on the Planning Analytics Workspace data tree.
6. To define a drill process with a DB Connection source, complete these steps:

   **Note:** A Satellite Connector connection for the current Planning Analytics database must be in place before you can see any DB Connection sources in the drill process editor. See Administer Satellite Connector for details on establishing a connection.

   a) Click **DB Connection**.
   b) Select the connection that you want to use. Optionally, click **Log in credentials** and supply a username and password for the connection.
   c) Click **Next**.
   d) Enter an SQL query to extract data from the source. The syntax and format of the query varies depending on which type of database you are accessing. If the query references a table name that contains spaces, you must enclose the name in double quotation marks.
   e) Click **Test and preview**.

      The process editor displays the first 10 records from the data source.

If the query doesn't return the results that you want, you can modify the query and click **Test query** to refine the results.

f) Click **Validate**.

If validation fails, see Troubleshooting.

g) Click **Save**.

TM1 saves the drill process as a TurboIntegrator process, but prefixes the name you assigned in step 8 with the string }Drill_. For example, if you save a drill process with the name PriceCubeToODBCSource, TM1 saves the process as }Drill_PriceCubeToODBCSource. Drill processes appear under the Control Objects for your database on the Planning Analytics Workspace data tree.

## Results

The drill process is now available to be included in a drill rule.

Troubleshooting:

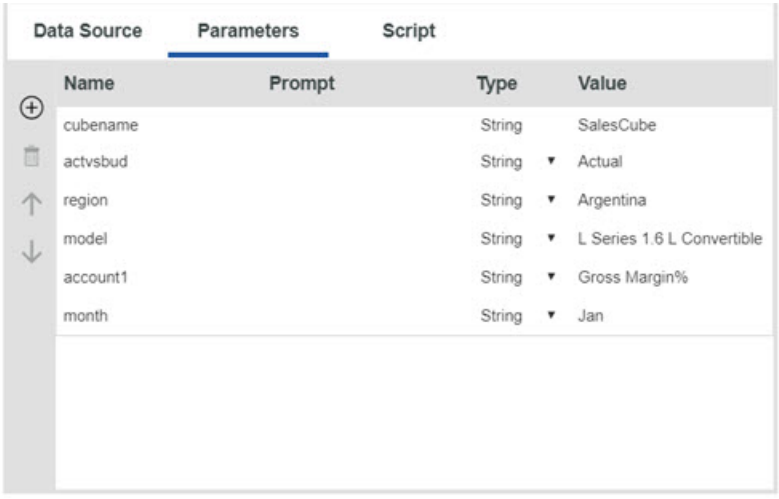
If any of the variables in your process have the same name as any of the parameters for the process, process validation fails with an Invalid Variable error.

A drill process creates variables and parameters for each dimension in the origination cube.

Variable names, which are created based on the content of your data source, appear in each column on the **Data Source** tab. In the following image, sandboxes, scenario, location, model, date, and value are all variables.




Parameter names appear in the Name column on the Parameters tab.




Parameter names are derived from the dimension names in the origination cube.

If validation fails due to identical naming, review the variable names and parameter names for your process. Modify any variable names that are identical to parameter names, then click **Validate** again.

## Edit a drill process

Planning Analytics Workspace modelers and administrators can edit drill processes.

### Procedure

1. In edit mode, go to the database in the Data tree where you want to edit a drill process, and expand the database to reveal the cubes.
2. Right-click the cube for which you want to edit a drill process.
3. Click **Drill**, **Edit drill process**.
4. Select the drill process, then click **Edit**.

   Use the drill process editor to modify the process.

## Delete a drill process

Planning Analytics Workspace modelers and administrators can delete drill processes.

### Procedure

1. In edit mode, go to the database in the Data tree where you want to delete a drill process, and expand the database to reveal the cubes.
2. Right-click the cube for which you want to delete a drill process.
3. Click **Drill**, **Delete drill process**.
4. Select the drill process, then click **Delete**.

# Object security

Modelers manage the security for cubes, dimensions, and processes in the settings editor, and security for dimension members in the security view in the dimension editor. You can assign security to these objects for any non-administrative user group in Planning Analytics Workspace.

You can also secure cells. Cell level security overrides all other Planning Analytics Workspace security.

To configure security, do the following steps:

1. Create user groups in the dimension editor.
2. Add users to the groups. Users are added to the groups in the **}ClientGroups** cube, which is in the **Control Objects** node of the data tree. You can also use TurboIntegrator processes to add users to groups, see AssignClientToGroup.
3. Define the access for cubes, dimensions, and processes for each group in the settings editor.
4. Define the access for dimension members in the dimension editor.
5. Define the access to cells if required.

Users are invited to IBM Planning Analytics Workspace by a Planning Analytics administrator.

**Related topics:**

Managing Users and Groups

IBM Business Analytics Proven Practices: How to implement element or cube-based security for IBM Cognos TM1

## Create a user group

You manage security in Planning Analytics Workspace by organizing users into groups.

Two types of user groups exist.

**Administrative Groups**
Predefined groups of ADMIN, DataAdmin, SecurityAdmin, and OperationsAdmin. You can't modify these groups. For more information, see the TM1 Operations guide, Administrative groups and authority.

**User Groups**
Includes all user-created groups.

You create user groups in the **Settings editor** or in the security view in the dimension editor. You can also create user groups by using the process ADDGROUP.

### Procedure

1. In edit mode, go to the database in the Data tree where you want to create a user group and expand the cubes.
2. Right-click **Cubes** and click **Edit settings**
3. To create a user group, click .
   The **Create New User Group** dialog box opens.
4. Enter the group name and click **Create**.

### What to do next

Next, add users to the user groups. Then, define the access for the user groups to dimension members, cubes, dimensions, processes, chores, and cells as needed.

## Add users to user groups

Modelers can add users to user groups in with a graphical interface of the **}ClientGroups** control cube.

Before users can be added to user groups, they must first be added to Planning Analytics Workspace by an administrator. To find out more, see “Upload a file to add or remove multiple users (cloud only)” on page 878.

**Tip:** You can also use processes to add users to user groups with the AddClient and AssignClientToGroup process functions.

### Procedure

1. Open a modeling workbench.
2. On the **Databases** tree, right-click the database where you want to add users to groups.
3. Click **Manage users and groups**.

   The **Users and groups** page displays a view with users on the rows and user groups on the columns.

| | ADMIN | 10000 | 10100 | 10110 | BUDGET VIEWER | BUDGET REVIEWER | BUDGET PROCESS A |
|---|---|---|---|---|---|---|---|
| Admin | ✓ | ✓ | ✓ | ✓ | ✓ | ✓ | ✓ |
| Reviewer | – | – | – | – | – | ✓ | – |
| Planner | – | – | – | – | – | ✓ | – |
| Goalsetter | – | ✓ | – | – | – | – | – |
| Budgetadmin | ✓ | ✓ | ✓ | ✓ | ✓ | ✓ | ✓ |
| Deeds | – | – | – | – | – | ✓ | – |
| Jones | – | – | ✓ | – | – | ✓ | – |
| Smith | – | – | – | ✓ | – | – | – |
| Howell | – | – | – | – | – | – | – |
| Bliss | – | – | – | – | – | ✓ | – |
| CAMID("pans:u:pa_prestgqc4_modeler... | ✓ | – | – | – | ✓ | ✓ | ✓ |
| CAMID("pans:u:pa_prestgqc4_consume... | ✓ | – | – | – | – | – | – |
| CAMID("pans:u:pa_prestgqc4_analyst0... | ✓ | – | – | – | – | – | – |
| CAMID("pans:u:pa_prestgqc4_admin01... | ✓ | – | – | – | – | – | – |
| CAMID("pans:u:pa_stgqc133_admin01... | – | – | – | ✓ | ✓ | ✓ | ✓ |
| CAMID("pans:u:pa_stgqc133_modeler0... | ✓ | – | – | – | – | – | – |

4. Right-click the cell at the intersection of a user and the group to which you want to assign the user, then click **Assign user(s) to groups(s)**.

**Tip:** You can use **SHIFT-click** and **CRTL-click** to select multiple cells. You can also click a row or column heading to select all cells in the row or column.

5. To unassign a user from a group, right-click the cell at the intersection of a user and the group, then click **Unassign user(s) to groups(s)**

# Secure dimension members

You manage security for dimension members in the dimension editor.

By default, all groups that have access to the database have write access to the dimension members. The first time that you open the security pane, a message is displayed: `Member security is presently disabled.` If you enable security for dimension members, the default access is removed, and you must set permissions for the user groups. If new members are later added to a dimension, you must assign access permissions for the new member to the appropriate user groups.

To find out more about what happens when you enable security, see " The security cube" on page 852.

## Secure dimension members Planning Analytics Workspace

### About this task

Follow these steps to apply member security using the dimension editor in Planning Analytics Workspace.

### Procedure

1. In the **Databases** tree on a modeling workbench, right-click the dimension for which you want to set member security, then click **Edit dimension**.
2. Click the **Security** tab on the dimension editor to open the security pane.
3. If member security is not enabled on the dimension, click **Enable**.

   The **Security** pane includes a column for each user group. You assign security for each member according to user group.
4. For each cell in the **Security** pane, select the security level for the corresponding member and user group. You can also copy and paste security levels.

   To set security for an entire user group column, click the column header, then click the action icon on the toolbar and click **Set security**.

   The security levels are as follows:

   **NONE**

   Members of the user group cannot see the member in the subset editor or dimension editor, and cannot see the cells identified by the member when browsing a cube.

   **READ**

   Members of the user group can see the cells identified by the member, but cannot change their data.

   **WRITE**

   Members of the user group can read and update the cells identified by the member and edit attributes of the member.

   **RESERVE**

   Members of the user group have all privileges implied by Write permission and can also reserve the member to prevent other users from updating cube cells identified by the member. The reservation can be removed either by the user who reserved the member or by users who have Admin rights for the member. A reservation expires automatically when the reserving user disconnects from the database or when the database shuts down.

**LOCK**

Members of the user group have all privileges implied by Write permission and can also lock the member. When a member is locked, nobody can update cube cells identified by the member. The lock can be removed only by users who have Admin rights for the member. Locks stays in place after the remote server shuts down

**ADMIN**

Members of the user group have complete access to the member.

**Example**

Say that you have a Region dimension with the following members:

- World
  - Americas
    - Canada
    - United States
  - Europe
    - France
    - Germany
    - United Kingdom
    - Switzerland

You might have two user groups, Americas_group, Europe_group, and assign functions as follows:

**Americas_group**
Write access to Americas.

Read access to Europe.

**Europe_group**
Write access to Europe.

Read access to Americas.

## The security cube

Security settings for dimension members are stored in a security cube called }ElementSecurity_*dimension_name* cube.

The }ElementSecurity_*dimension_name* cube does not exist by default, and all users have access to the dimension members by default.

When you edit the security setting for a dimension for the first time in the dimension editor, or create a user group, you are prompted to create a security cube.

When the security cube is created, users must be given explicit write access to all of the members in the dimension.

The security cube contains the following dimensions:

***Dimension_name***
The dimension and members for which the security cube is created for.

**}Groups**
User groups that are defined for the TM1 database.

The }ElementSecurity_*dimension_name* cube is in the **Control Objects** > **Cubes** node of the data tree.

# Secure cubes, dimensions, processes, and chores

Modelers can manage security settings for cubes, dimensions, processes, and chores in the security editor.

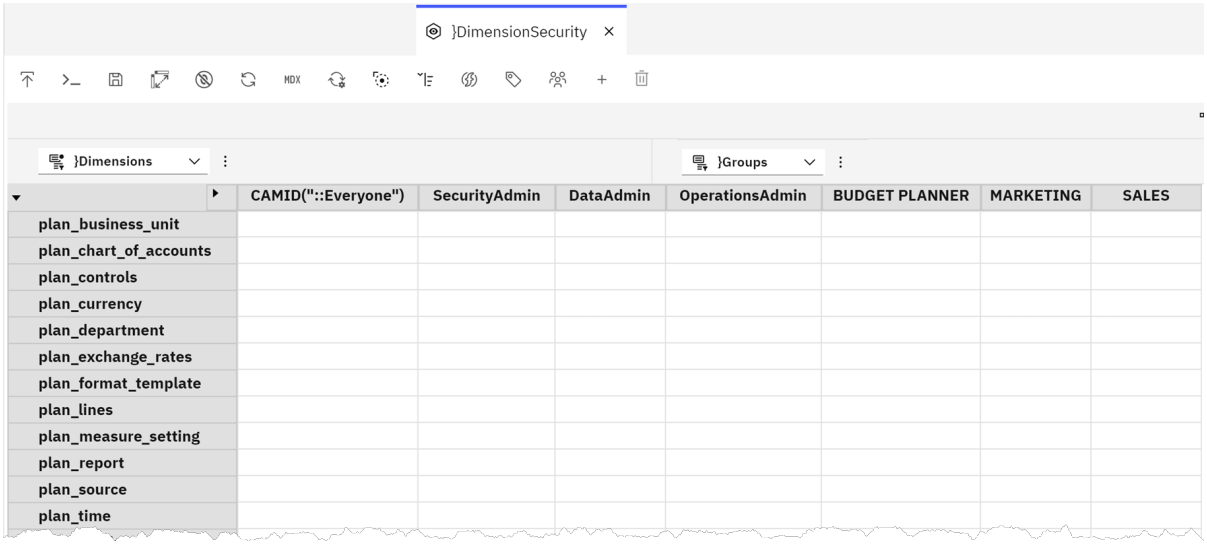
**Note:** You must assign security for dimension hierarchies independent of the parent dimension. Hierarchies do not inherit security from the parent dimension. Security for hierarchies must be set in the }DimensionSecurity control cube. For details, see "Manage security for dimension hierarchies" on page 854.

## Procedure

1. On a modeling workbench, expand the database for which you want to manage security on the **Databases** tree, then right-click either the**Cubes**, **Dimensions**, **Processes**, or **Chores** node.
2. Click **ObjectName Security**. For example, **Cube Security** or **Chore Security**.

   A modified view of the applicable control cube opens, with individual database objects on the row axis and user groups on the column axis.

   For example, here's the view of the `}DimensionSecurity` control cube you see when you click **Dimensions** > **Dimension Security**.




You can assign these security levels for objects:

**NONE**
: Members of the user group have no access to the object and cannot see or use the object. When NONE is selected, the cell is blank in the editor.

**READ**
: Members of the user group can view objects, but cannot write to the objects.

**WRITE**
: Members of the user group can view and update objects.

**RESERVE**
: Members of the user group can view and edit objects, and can temporarily reserve objects to prevent other users from updating them.

**LOCK**
: Members of the user group can view and edit objects, and can permanently lock objects to prevent all users (including the person who set the security) from updating them.

**ADMIN**

Members of the user group have complete access to objects.

Use these options to assign security for objects on the Planning Analytics database:

- To assign security for a single user group to a single object, double-click the cell at the intersection of the user group and object, then click the desired security level.
- To assign security to multiple adjacent cells, click the first cell and SHIFT+click the last cell, then right-click the last cell and click **Cell security**.
- To assign security to multiple nonadjacent cells, CTRL+click each cell, then right-click the last cell and click **Cell security**.
- To assign security for an entire row, right-click the element name on the row header, then click **Cell security**. You can also use CTRL+click or SHIFT+click to select multiple rows.
- To assign security for an entire column, right-click the element name on the column header, then click **Cell security**. You can also use CTRL+click or SHIFT+click to select multiple columns.

3. You can sort or filter the items in a row or column. Right-click a row or column header, then click either **Sort** or **Filter**.
4. Click + on the security editor to create a new group on the Planning Analytics database. You can then immediately apply object security to the new group.
5. (Cloud only) Click to import groups that exist in Planning Analytics Workspace to the Planning Analytics database.

   The groups you select are added to the database. In addition, any users within the groups are also added to the database and assigned to the appropriate groups. You can then immediately apply object security to the new groups.

   You cannot import groups that exist in Planning Analytics Workspace to the Planning Analytics database on Planning Analytics Workspace Local.

# Manage security for dimension hierarchies

As of Planning Analytics 2.0.9.5, you must assign security for dimension hierarchies independent of the parent dimension. Hierarchies do not inherit security from the parent dimension. If you do not explicitly set security for an hierarchy, the default security is NONE.

## About this task

Hierarchy security must be defined in the }DimensionSecurity control cube. Hierarchies are not exposed in the dimension settings editor.

The following rules apply to security on hierarchies.

- The hierarchy with the same name as the dimension shares the same security as the parent dimension. The same named hierarchy cannot have a different security level than the dimension.
- A user group cannot have higher security access to an hierarchy than to the parent dimension.
- Any security applied to the Leaves hierarchy is ignored. If a user has READ privilege on the parent dimension they will also have READ privilege to the Leaves hierarchy. The Leaves hierarchy is automatically updated to include all leaf members of all hierarchies in a dimension.
- READ privilege or higher is required to see both the subsets and members in a hierarchy.
- LOCK and RESERVE privilege are not applicable to hierarchies.

## Procedure

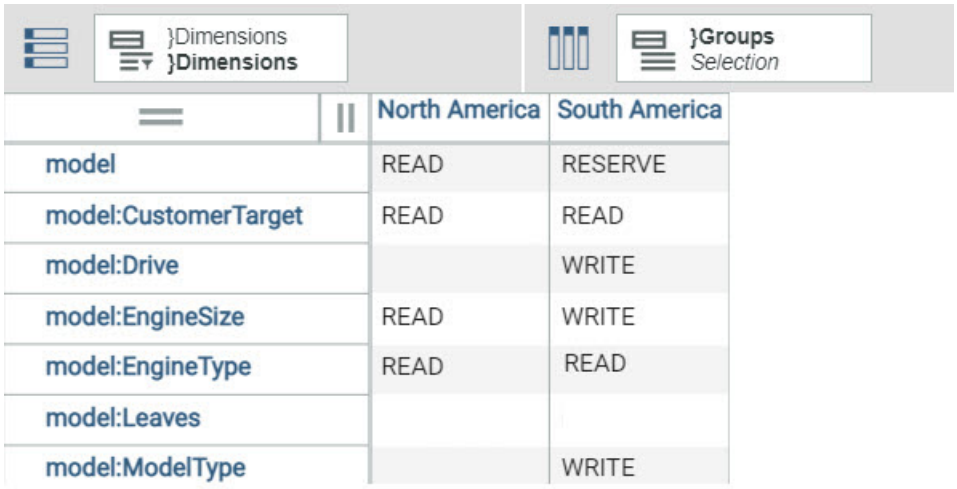
1. Open the }DimensionSecurity control cube.

   The control cube contains two control dimensions, }Dimensions and }Groups.

Hierarchies are prefixed with the parent dimension name, *dimension_name:hierarchy_name*. In this example, `model:CustomerTarget`, `model:Drive`, `model:EngineSize` and others are hierarchies of the `model` dimension.

2. Enter the desired security privilege at the intersection of an hierarchy and a user group. Security privileges for user groups are described in Assigning Security Rights to Groups.

   For example, here the South America user group has READ access to the `model:CustomerTarget` and `model:EngineType` hierarchies.




| | North America | South America |
|---|---|---|
| model | READ | RESERVE |
| model:CustomerTarget | READ | READ |
| model:Drive | | WRITE |
| model:EngineSize | READ | WRITE |
| model:EngineType | READ | READ |
| model:Leaves | | |
| model:ModelType | | WRITE |

## Secure cells

You can secure specific cells in aPlanning Analytics Workspace cube.

Cell-level security overrides all other security. If you want member-level security to apply, cell-level security for the cell must be undefined.

When you secure cells, a cell security control cube is created. The cell security control cube contains the dimensions of the cube to which you want to apply cell-level security. You can set security for cells in the control cube by assigning security permissions to user groups.

Cell-level security applies to leaf members and generally does not apply to consolidations, although you can use the None and Read security permissions to control the display or editing of consolidations.

### Before you begin

The user group that you are assigning access to must have access to the cube and the dimensions in the cube. See “Secure cubes, dimensions, processes, and chores” on page 853 to find out more.

### Procedure

1. To create cell security, right-click a cube on the **Databases** tree on a modeling workbench, then click **Create cell security**.

   If cell security is already defined for a cube, right-click a cube on the **Databases** tree on a modeling workbench, then click **Edit cell security** and proceed to Step 4.
2. Select the dimensions you want to include in the cell security cube.
3. Click **Create**.

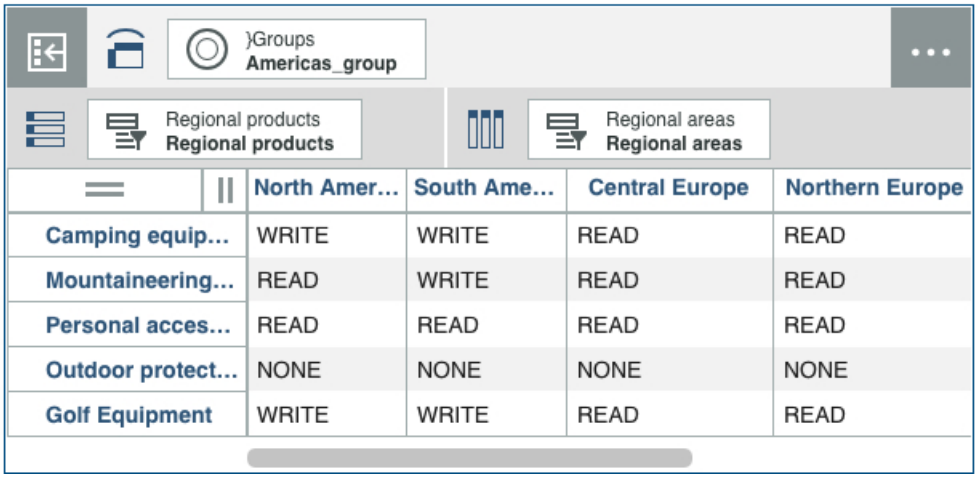
   A cell security view and the rule editor are added to the sheet. The cell security cube is created in **Control Objects** > **Cubes** and is called **}CellSecurity_*cubename***. You can create rules to assign cell security, or you can assign cell security directly in the cell security view.
4. To assign cell security in the rule editor, type the rule in the rule editor.

```
['member','security_group'] = S:'security_level'
```

The following example prevents any users in Europe_group from viewing any cells that are identified by the members pineapple and France.

```
['pineapple','France','Europe_group'] = S:'NONE';
```

5. To assign cell security in the cell security control cube, drag the **}Groups** view onto the context and select the user group that you want to assign security to.




6. Double-click the cell to which you want to assign security, then click the desired security level.

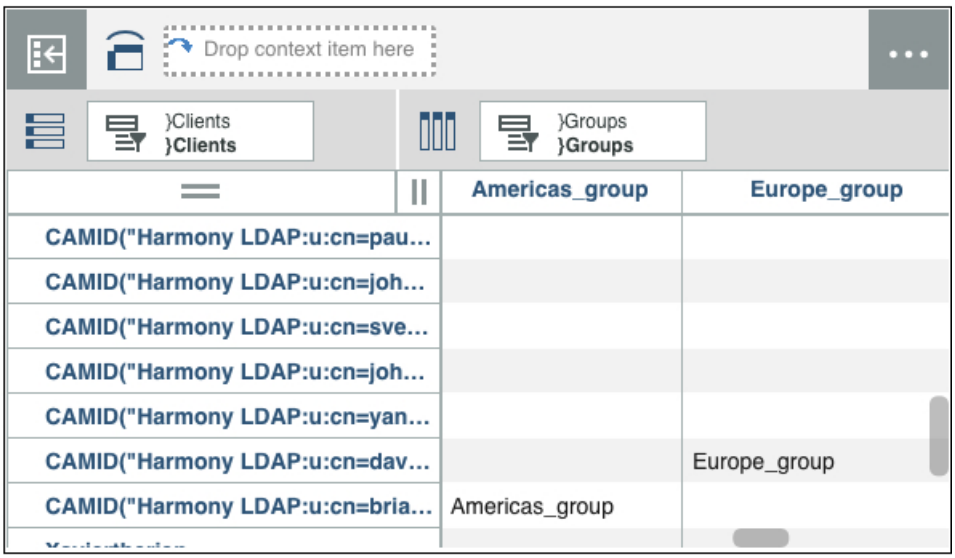
## Cell security example

This example starts with a simple Products cube with a regional areas dimension and regional products dimension.

Some product ranges are only sold in some regions, so you want to prevent data from being entered for these regions.

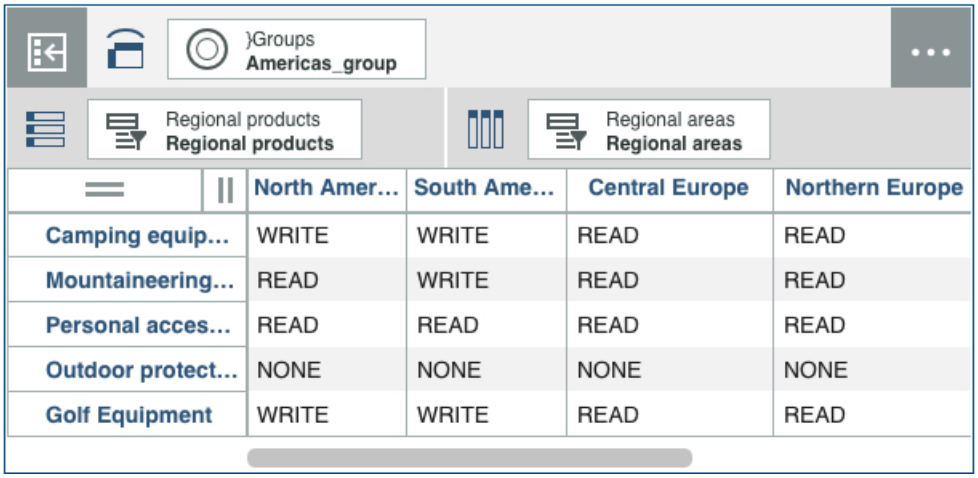
The steps to prevent data from being entered for product ranges in some regions are as follows.

1. You are logged on to Planning Analytics Workspace as an administrator. Create a user group in the Security tab of the dimension editor, and name it **Americas_group**.
2. In the **}ClientGroups** cube, add the sales manager for the Americas territory to the **Americas_group**.



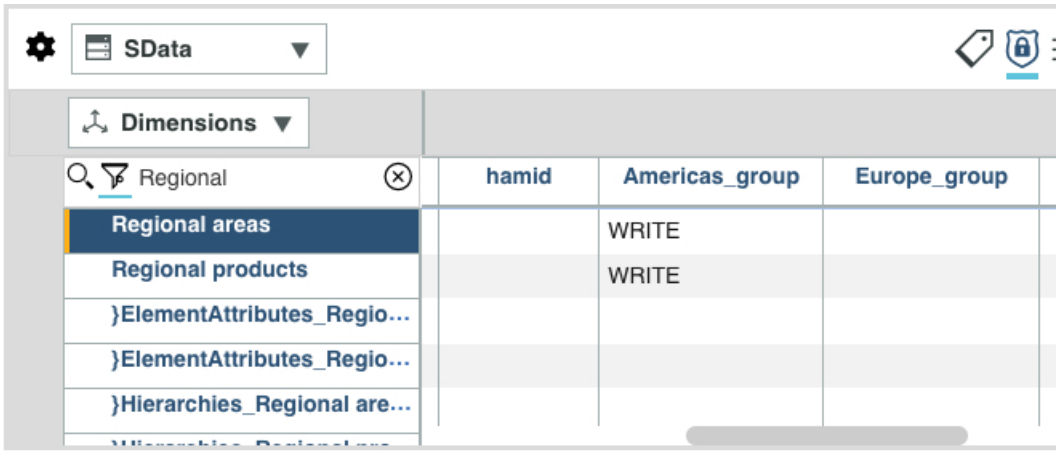

3. In the Data tree, right-click the Regional products cube, and select **Edit cell security** . Both the rules editor and the cell security views are opened on the sheet. You can either set up the security by using rules, or you can set the security in the cell security view. This example uses the cell security view.

4. In the cell security view, drag the **}Groups** dimension onto the context in the overview and find the **Americas_group**. Assign some different security to the cells for the **Americas_group** so that you can verify that the security is set up correctly. For example, a mixture of WRITE, READ, and NONE.




| | North Amer... | South Ame... | Central Europe | Northern Europe |
|---|---|---|---|---|
| Camping equip... | WRITE | WRITE | READ | READ |
| Mountaineering... | READ | WRITE | READ | READ |
| Personal acces... | READ | READ | READ | READ |
| Outdoor protect... | NONE | NONE | NONE | NONE |
| Golf Equipment | WRITE | WRITE | READ | READ |

5. In the Data tree, go to the database and right-click **Cubes** > **Edit settings** to open the Settings editor and give WRITE access to the Regional products cube for **Americas_group**. Then, select **Dimensions** from the **Settings editor** menu, and give WRITE access to the Regional areas and Regional products dimension for the **Americas_group**.




6. Add the Regional products cube view to the sheet and save the view as a separate object. Click the view to display the toolbar, click **Save**  , then select **Save as** to specify a name.
7. Enter some numbers into the view.

| | North Amer... | South Ame... | Central Eur... | Northern E... | Southern E... |
|---|---|---|---|---|---|
| Camping equip... | 657,210 | 546,790 | 89,202 | 245,998 | 478,390 |
| Mountaineering... | 1,245,690 | 734,009 | 456,283 | 128,530 | 297,845 |
| Personal acces... | 789,003 | 765,112 | 58,390 | 322,290 | 322,504 |
| Outdoor protect... | 56,700 | 78,963 | 846,711 | 85,732 | 837,632 |
| Golf Equipment | 899,881 | 243,522 | 98,712 | 492,842 | 42,998 |

8. Open a different browser, and log on as the Americas sales manager who is a member of the **Americas_group**, and is not a member of any other group.
9. Open the Regional products view.

| | North America | South America | Central Eur… | Northern Eu… | Southern Eu… |
|---|---|---|---|---|---|
| Camping equipm… | 657,210 | 546,790 | 89,202 | 245,998 | 478,390 |
| Mountaineering e… | 1,245,690 | 734,009 | 456,283 | 128,530 | 297,845 |
| Personal accessories | 789,003 | 765,112 | 58,390 | 322,290 | 322,504 |
| Outdoor protection | | | | | |
| Golf Equipment | 899,881 | 243,522 | 98,712 | 492,842 | 42,998 |

In the example shown, you can see that the Americas sales manager can see nothing for Outdoor protection products. Other cells shaded dark gray are readable but not writable, and cells with a white or light gray background are writable.

## Delete a user group

You can delete user groups in the security view in the **Dimension Editor** when you create or edit members of a hierarchy.

### Procedure

1. To open the security pane, in the **Dimension Editor**, click  .
2. To delete a user group, select the user group and click  .
3. Click **Ok** to confirm that you want to remove the selected user group.

# Localize a model

You can localize a model in Planning Analytics Workspace by creating translated values for cubes, dimensions, members, views, processes, and public sets.

## Localize cubes, dimensions, processes, or chores

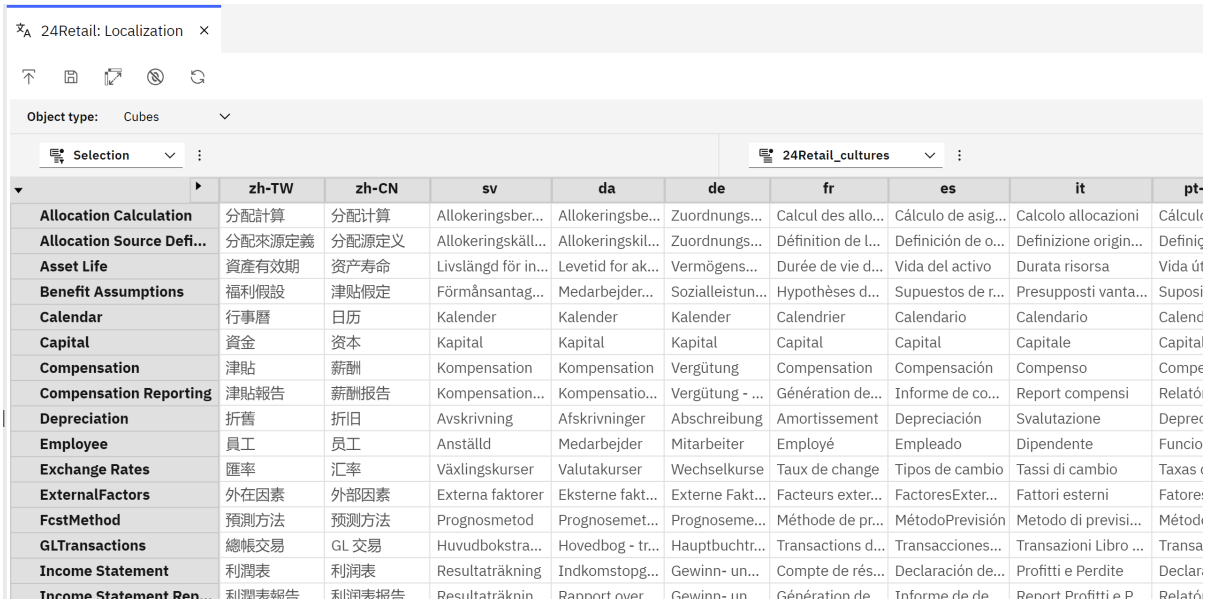
The Localization editor makes it easier to localize the names of cubes, dimensions, members, processes, and chores on your Planning Analytics Workspace database.

The Localization editor is available from the modeling workbench.

To localize cubes, dimensions, processes, or chores, right-click a database on the **Databases** tree, then click **Localization** > **object type**.

You can also open the Localization editor by right-clicking an object node on the Databases tree, then clicking **Localization**. For example, **Cubes** > **Localization** or **Chores** > **Localization**.

The Localization editor displays a grid, with object names on the rows and the languages included in the }Cultures control dimension on the columns.




| | zh-TW | zh-CN | sv | da | de | fr | es | it | pt- |
|---|---|---|---|---|---|---|---|---|---|
| Allocation Calculation | 分配計算 | 分配计算 | Allokeringsber... | Allokeringsbe... | Zuordnungs... | Calcul des allo... | Cálculo de asig... | Calcolo allocazioni | Cálcul |
| Allocation Source Defi... | 分配來源定義 | 分配源定义 | Allokeringskäll... | Allokeringskil... | Zuordnungs... | Définition de l... | Definición de o... | Definizione origin... | Definiç |
| Asset Life | 資產有效期 | 资产寿命 | Livslängd för in... | Levetid for ak... | Vermögens... | Durée de vie d... | Vida del activo | Durata risorsa | Vida út |
| Benefit Assumptions | 福利假設 | 津贴假定 | Förmånsantag... | Medarbejder... | Sozialleistun... | Hypothèses d... | Supuestos de r... | Presupposti vanta... | Suposi |
| Calendar | 行事曆 | 日历 | Kalender | Kalender | Kalender | Calendrier | Calendario | Calendario | Calend |
| Capital | 資金 | 资本 | Kapital | Kapital | Kapital | Capital | Capital | Capitale | Capital |
| Compensation | 津貼 | 薪酬 | Kompensation | Kompensation | Vergütung | Compensation | Compensación | Compenso | Compe |
| Compensation Reporting | 津貼報告 | 薪酬报告 | Kompensation... | Kompensatio... | Vergütung - ... | Génération de... | Informe de co... | Report compensi | Relató |
| Depreciation | 折舊 | 折旧 | Avskrivning | Afskrivninger | Abschreibung | Amortissement | Depreciación | Svalutazione | Deprec |
| Employee | 員工 | 员工 | Anställd | Medarbejder | Mitarbeiter | Employé | Empleado | Dipendente | Funcio |
| Exchange Rates | 匯率 | 汇率 | Växlingskurser | Valutakurser | Wechselkurse | Taux de change | Tipos de cambio | Tassi di cambio | Taxas |
| ExternalFactors | 外在因素 | 外部因素 | Externa faktorer | Eksterne fakt... | Externe Fakt... | Facteurs exter... | FactoresExter... | Fattori esterni | Fatores |
| FcstMethod | 預測方法 | 预测方法 | Prognosmetod | Prognosemet... | Prognoseme... | Méthode de pr... | MétodoPrevisión | Metodo di previsi... | Método |
| GLTransactions | 總帳交易 | GL 交易 | Huvudbokstra... | Hovedbog - tr... | Hauptbuchtr... | Transactions d... | Transacciones... | Transazioni Libro ... | Transa |
| Income Statement | 利潤表 | 利润表 | Resultaträkning | Indkomstopg... | Gewinn- un... | Compte de rés... | Declaración de... | Profitti e Perdite | Declar |
| Income Statement Rep... | 利潤表報告 | 利润表报告 | Resultaträknin... | Rapport over... | Gewinn- un... | Génération de... | Informe de de... | Report Profitti e P... | Relató |

You can click the **Object type** menu and select a different object type to localize.

The first time you open the Localization editor for an object type, a new localization control cube is created on the database where the objects reside. Localized object names are saved in these control cubes.

| Object type | Localization control cube |
|---|---|
| Cubes | }LocalizedCubeAttributes |
| Dimension | }LocalizedDimensionAttributes |
| Processes | }LocalizedProcessAttributes |
| Chores | }LocalizedChoresAttributes |

To localize object names, you can work directly in the Localization editor or upload data to the appropriate localization control cube.

**Localize object names in the Localization editor**
Click the cell at the intersection of an object name and a language, then enter the appropriate localized name.

**Upload data to a localization control cube**
Right-click the appropriate localization control cube on the **Databases** tree, then click **Import data**. See Importing data into a cube for more information.

## Localize dimension members

The Localization editor makes it easier to localize the names of dimension members on your Planning Analytics Workspace database.

The Localization editor is available from the modeling workbench.

To localize dimension member names, right-click an individual dimension on the **Databases** tree, then click **Localization**.

The Localization editor displays a grid, with member names on the rows and the languages included in the }Cultures control dimension on the columns.

The first time you open the Localization editor for members in a specific dimension, a new localization control cube is created on the database where the dimension resides. The localization control cube is named `}LocalizedElementAttributes_`*`dimName`*. Localized dimension member names are saved in this control cube.

For example, when you first open the Region dimension in the Localization editor, the }LocalizedElementAttributes_Region control cube is created.

To localize dimension member names, you can work directly in the Localization editor or upload data to a }LocalizedElementAttributes_*dimName* control cube.

**Localize member names in the Localization editor**
Click the cell at the intersection of a dimension member name and a language, then enter the appropriate localized name.

**Upload data to a localization control cube**
Right-click the appropriate }LocalizedElementAttributes_*dimName* localization control cube on the **Databases** tree, then click **Import data**. See Importing data into a cube for more information.

# Localize set names

To display set names in other languages, create a process that creates the Caption attribute for all member names in the dimension, and then assigns values for the set names that you want to translate. This option is available only if your organisation has deployed IBM Planning Analytics version 2.0.4 or later.

## About this task

A Caption attribute is a type of attribute that is required for every set that you want to translate.

Caption attributes are created through the TurboIntegrator process. When you create Caption attributes, they must be set up as an alias type with unique locale-specific values.

Planning Analytics Workspace then uses the browser language settings to determine which language to display. You can see objects in that language without needing any additional configuration.

International language codes that are defined by ISO 639-1 are used to identify major languages and IETF language tags to identify specific locales. For example, fr identifies French, while fr-CA identifies French (Canada).

You can assign Caption attribute values for major language codes, such as fr, as well as any associated specific locales, such as fr-Fr or fr-CA.

If a Caption attribute value does not exist for a given specific locale, the value of the associated major language code is retrieved. For example, if a Caption attribute value does not exist for pt-BR, the value for pt is retrieved.

If no values are found for a Caption attribute, the base default attribute value is returned.

You can see which ISO 639-1/IETF combinations are supported in the }Cultures control dimension.

## Procedure

1. Create a new process.
2. In the Prolog, create the Caption attribute as an alias attribute.
   Type
   ```
   SubsetAttrInsert('<dim_name>', '', 'Caption', 'A');
   ```
   To find out more, see SubsetAttrInsert.
   This function creates a control cube called SubsetAttributes_*<dim_name>* with the following dimensions: }Subsets_*<dim_name>*, }SubsetAttributes_*<dim_name>*.
3. For each subset that you want to translate, add a SubsetAttrPutS function for each language that you want to make available on your database:
   ```
   SubsetAttrPutS( '<String>', '<dim_name>', '<set_name>', 'Caption', '<Lang_Locale_Code>' );
   ```

For example, if you want to display the Sales set in the Departments dimension in French, include the following function:

```
SubsetAttrPutS( 'Ventes', 'Departments', 'Sales', 'Caption', 'fr' );
```

This creates a control cube called LocalizedSubsetAttributes_*<dim_name>* with the following dimensions: }Subsets_*<dim_name>*, }Cultures, }SubsetAttributes_*<dim_name>*.

### What to do next

To test this process, save the book, then set the browser language setting to the language that you want to display. Refresh the browser and add a view that contains the translated set. Check that the set name is displayed in the correct language.

## Localize view names

To display view names in other languages, create a process that creates the Caption attribute for all views in a cube. Then, assign values for the view names that you want to translate.
This option is available only if your organization deployed IBM Planning Analytics version 2.0.4 or later.

**Note:** View names can be translated only for views that are saved in the database. Views that are created in IBM Planning Analytics Workspace are not saved in the database. They are saved in the book.

### About this task

A `Caption` attribute is a type of attribute that is required for every view that you want to translate.

`Caption` attributes are created through the TurboIntegrator process. When you create `Caption` attributes, they must be set up as an alias type with unique locale-specific values.

Planning Analytics Workspace then uses the browser language settings to determine which language to display. You can see objects in that language without needing any additional configuration.

International language codes that are defined by ISO 639-1 are used to identify major languages and IETF language tags to identify specific locales. For example, `fr` identifies French, while `fr-CA` identifies French (Canada).

You can assign `Caption` attribute values for major language codes, such as `fr`, as well as any associated specific locales, such as `fr-Fr` or `fr-CA`.

If a `Caption` attribute value does not exist for a given specific locale, the value of the associated major language code is retrieved. For example, if a `Caption` attribute value does not exist for `pt-BR`, the value for `pt` is retrieved.

If no values are found for a `Caption` attribute, the base default attribute value is returned.

You can see which ISO 639-1/IETF combinations are supported in the }Cultures control dimension.

### Procedure

1. Create a new process.
2. Click **Script**, and in the **Prolog**, create the Caption attribute as an alias attribute.
   Type
   ```
   ViewAttrInsert('<cube_name>', '', 'Caption', 'A');
   ```
   This function creates a control cube that is called ViewAttributes_*<cube_name>* with the following dimensions: }Views_*<cube_name>*, }ViewAttributes_*<cube_name>*. To find out more, see ViewAttrInsert.
3. For each view that you want to translate, add a ViewAttrPutS function for each language that you want to make available on your database:

```
ViewAttrPutS( '<String>', '<cube_name>', '<view_name>', 'Caption', '<Lang_Locale_Code>');
```

For example, to translate a view called Price into French on a cube that is called SalesCube, include the following function:

```
ViewAttrPutS( 'Prix', 'SalesCube', 'Price', 'Caption', 'fr' );
```

This function creates a control cube that is called LocalizedViewAttributes_*<cube_name>* with the following dimensions: }Views_*<cube_name>*, }Cultures, }ViewAttributes_*<cube_name>*.

### What to do next

To test this process, save the book, then set the browser language to the language that you want to display. Refresh the browser and find the view with the translated name in the data tree. Check that the view name is displayed in the correct language.

## Manage Planning Analytics Workspace database log files

You can view database transaction logs, message logs, and audit logs from a Modeling workbench.

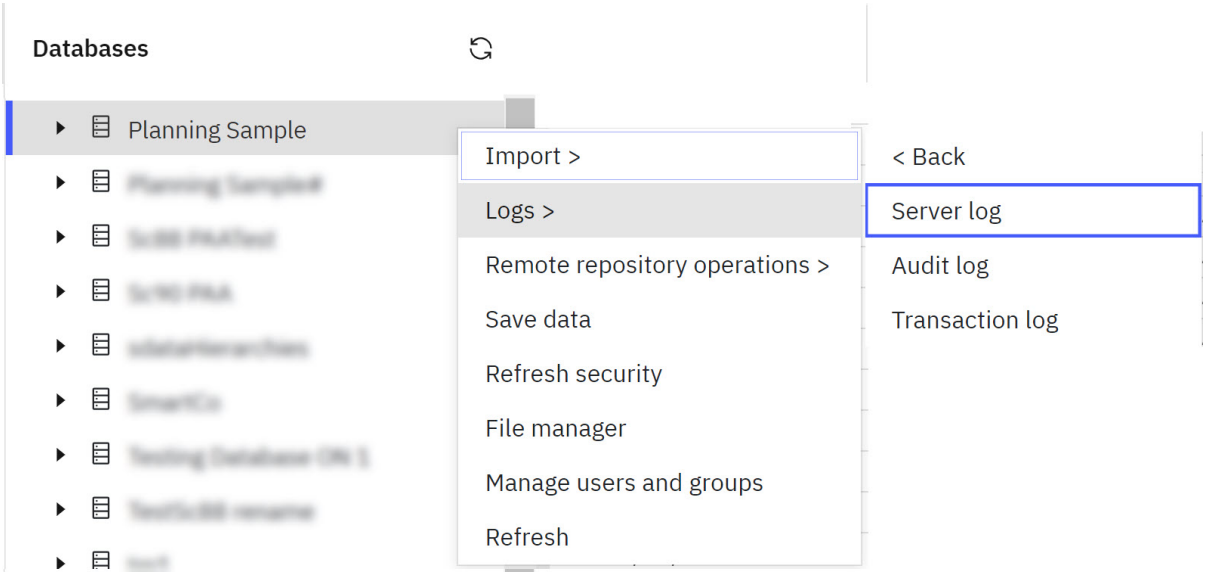
### About this task

Access to server logs is available only on the modeling workbench. You cannot open logs from a book.

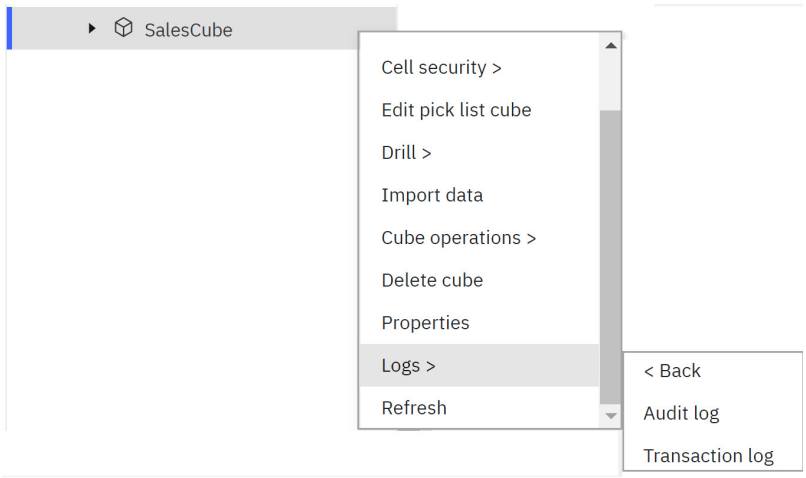
The option to open the server log and the transaction log is available for all databases. To access the audit log, the **Audit log** configuration parameter must be enabled for the database.

### Procedure

1. To open any of the available database logs, open a modeling workbench.
2. Right-click the database on the **Databases** tree.
3. Click **Logs**.
4. Click the log that you want to open.




The audit log and transaction log can also be opened when you right-click an individual cube on the **Databases** tree.




# Server log

## About this task

The server log (`tm1server.log` file) captures messages that are issued on an individual Planning Analytics database. Each entry in the log includes this information:

- The ID number of the thread on the database that generated the message.
- The severity level of the message: Debug, Info, Warn, Error, or unrecoverable.
- The name of the Planning Analytics component that generated the message.
- A text message that describes the error or activity.

The message log is automatically managed by the Planning Analytics database. By default, the maximum message log file size is 100 MB. When the log file grows larger than 100 MB, it is rolled over automatically by the Planning Analytics database. The `tm1server.log` file is renamed `tm1server.log.1`, and an empty `tm1server.log` file is created.

By default, a maximum of 20 Planning Analytics database message log files are maintained by the Planning Analytics database (`tm1server.log, tm1server.log.1,...tm1server.log.19`). Maximum log file size and maximum number of historical files are based on configuration settings in the `tm1s-log.properties` file.

The order of reading the logs when multiple log files are present should be the current `tm1server.log` file, then the highest .# entry to the lowest. For example, if four log files are present, the order would be `tm1server.log, tm1server.log.3, tm1server.log.2, tm1server.log.1`. You can also consult the time stamp for the log files and read them from most recent to oldest to make sure you are reviewing items in the correct order.

## Procedure

1. The server log is live and updated in real-time. You might want to pause the display of updates while you are examining messages. Click **Pause** to temporarily pause updates in the message log viewer, then click **Resume** to continue updates. Pausing updates only affects the display of messages in the log viewer; messages continue to be logged and are displayed when you resume updates.
2. The server log contains information about process execution. When a process encounters an error and generates a process error log, you can open the process error log directly from the server log. To open the process error log, click the process error log filename in the server log. Process error logs use the file naming convention TM1ProcessError_*timestamp_processname*.log.
3. By default, the log viewer displays messages for the current day. You can click to reveal the time period selector. Click a different time period and the click **Apply** to view messages for a 1 day, 7 day, 30 day, or custom time period.

4. Click the **Download** icon to download the server log as a text file.
5. Click the **Settings** icon to set view options for the log.

# Audit log

## About this task

The audit log monitors changes to metadata, such as modifications to dimensions, views, and subsets. It also tracks login and TurboIntegrator process execution.

**Note:** The audit log is not available when using Planning Analytics Workspace connected to TM1 Database 12.

This log can be used for accountability purposes where laws or regulations require auditing of certain activities in mission-critical software. Administrators and modelers can use an audit log to answer these questions:

- What object was changed?
- How was the object modified?
- Who made the change?
- When was the change made?

You can open an audit log both from a **Database** node or an individual cube on the **Databases** tree. If you open from a **Database** node, the log displays details for all objects on the database. If you open the log for an individual cube, the **Filter** option on the log is set to display details only for the selected cube.

## Procedure

1. By default, the log viewer displays messages for the current day. To view messages for a different time period, select a period from the **Time period** menu, then click **Search**.
2. Click the **Filter** icon to filter by user, description, object type, or object name.
3. Click the **Download** icon to download the audit log as a .csv file.
4. Click the **Settings** icon to set view options for the log.

# Transaction log

## About this task

The following information is recorded about data changes in the transaction log:

- Date and time the change was made.
- The name of the client who made the change.
- The name of the cube in which the change was made.
- The cell members that identify the cell that changed.
- The value before the change .
- The value after the change .
- The change set that contains the change.

By default, the log viewer displays transactions for the current day, all users, and all cubes.

## Procedure

1. To view transactions for a different time period, select a period from the **Time period** menu, then click **Search**.
2. To view transactions for a single user, enter the full **User** name, then click **Search**.

3. To view transactions for a single cube, select a cube from the **Cube** menu, then click **Search**.

   You can filter on one or more members in the selected cube.

   a) Click the **Filter** icon.

   b) Select the dimension members for which you want to view transactions.

   c) Click **Done**.
4. Click the **Download** icon to download the log as a .csv file.
5. Click the **Settings** icon to set view options for the log.

## Roll back transactions in the transaction log

You can roll back one or more transactions from the database transaction log in Planning Analytics Workspace.

### Procedure

1. Open the transaction log from the **Databases** tree on a modeling workbench.
   a) To open the transaction log at the database level, right-click a database, then click **Logs** > **Transaction log**.
   b) To open the transaction log for an individual cube, right-click the cube, then click **Logs** > **Transaction log**.
2. Set filtering options if desired, then click **Search**. You can also click any column heading to sort transactions.
3. Select the transactions you want to revert. You can roll back up to 1500 transactions in a single roll back action.
4. Click **Roll back** to roll back a single selected transaction or click **Roll back all** to roll back multiple selected transactions.
5. Click **OK** to confirm the action.

   The current cell value is reverted to the value indicated by the **Prior value** column. If multiple selections target the same cell, the most recent transaction takes precedence.

# Manage database threads on the modeling workbench

You can view and manage thread activity for a database directly on the modeling workbench.

This feature provides many of the same capabilities available for thread monitoring in Planning Analytics Administration.

To monitor thread activity on a Planning Analytics database, right-click the database on the **Databases** tree, then click **Threads**.

The thread monitor displays all threads on the database using these sortable columns. The availability of columns varies between Planning Analytics database and TM1 Database 12. You can click **Settings** to manage the display of columns.

**Name**
The name of the user who initiated the thread.

**State**
The current thread state.

- Idle: not using resources
- Wait: waiting on other threads, blocked
- Running
- Commit: committing updates made by a thread
- Rollback: rolling back to a state before an update

- Login: when a user logs in and initiates a thread

**Type**
Indicates a User or System thread.

**Function**
The API function being executed by the thread.

**Wait**
The length of time, in seconds, that the thread has been in a wait state.

**Elapsed**
The total elapsed time of the thread in its current state.

**W/R/Ix locks**
The number of write, read, and intent-to-write locks that the thread holds.

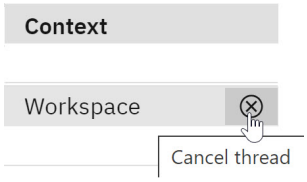
A write lock allows only the thread that holds the lock to access and write changes to an object. No other thread can read or modify the object until the W lock has been released.

A read lock allows many threads to read from an object at the same time, but does not allow another thread to modify or write to the object until all R locks have been released.

An intent-to-write lock reserves the right for a thread to obtain a write lock on an object when all read locks have been released. Only one thread at a time is allowed to have an Ix lock on an object.

**Context**
The connected Planning Analytics client that is associated with the thread.

To cancel an individual thread, click the **Cancel** icon adjacent to the final column on the report, then confirm the action.




To cancel multiple threads:

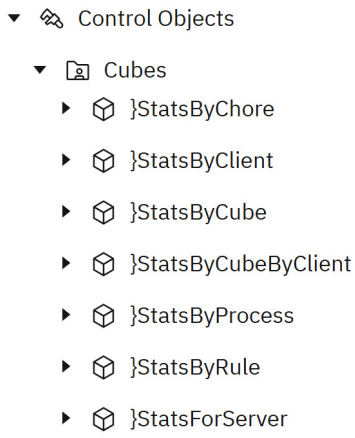
1. Click **Pause**. When monitoring is paused, check boxes are available to select individual threads.
2. Select the threads you want to cancel.
3. Click **Cancel threads** on the toolbar, then confirm the action.
4. Click **Resume** to resume thread monitoring.

# Start and stop performance monitoring from the modeling workbench (TM1 Database 11 only)

You can start or stop performance monitoring for a database from the modeling workbench in Planning Analytics Workspace on TM1 Database 11.

## About this task

When performance monitoring is enabled, performance and usage statistics are written to the `}Stats` control cubes for the database. You can use these control cubes to review minute-by-minute statistics for chores, clients, cubes, processes, rules, and the database (server).




### Procedure

1. On the **Databases** tree, click the **Action** menu for a database.
2. Click **Performance monitor**.
3. Enable or disable the **PerformanceMonitorOn** option.
4. Click **Save**.

### Results

The setting of the **PerformanceMonitorOn** option on the modeling workbench determines the status of the `PerformanceMonitorOn` configuration parameter for the database. Administrators can continue to manage the `PerformanceMonitorOn` configuration parameter in Planning Analytics Administration. The most recent setting for the `PerformanceMonitorOn` configuration parameter takes precedence, whether set on the modeling workbench or in Planning Analytics Administration.

## Use Git repository integration to manage Planning Analytics database objects

Modelers can access a Git repository directly from Planning Analytics Workspace. You can use the Git repository to store Planning Analytics database objects such as cubes, dimensions, rules, and processes that are common across multiple models.

You can also use the repository to move database objects from one environment to another. For example, you might want to copy database objects from a development environment to a production environment.

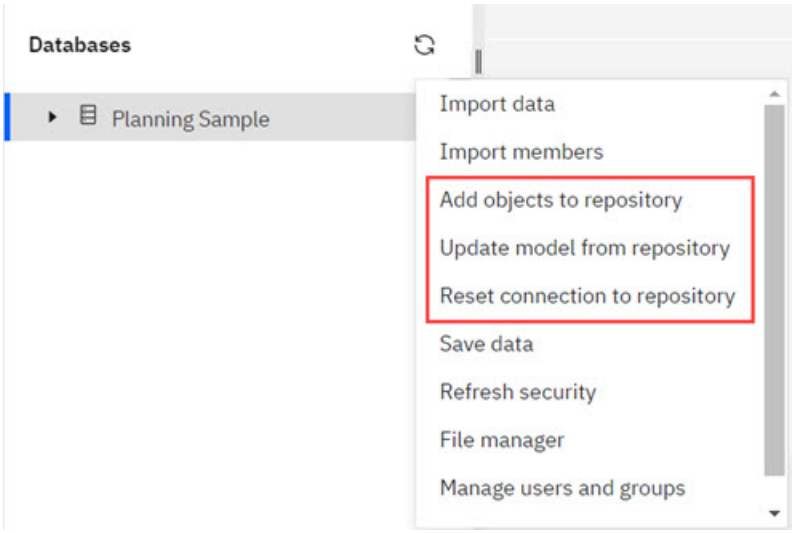
The repository can also serve as a version control facility for your Planning Analytics database objects.

Git repository integration on Planning Analytics Local and Planning Analytics on Cloud requires Planning Analytics (TM1) database version 11.5 or later.

Repository integration is supported on these three Git providers:

- Microsoft Azure (https://azure.microsoft.com/services/devops/repos/)
- AWS CodeCommit (https://aws.amazon.com/codecommit/)
- GitHub (https://github.com)

Options to interact with a Git repository are available when you click the **Action** icon for a database on the **Databases Tree** of a modeling workbench.




The following limitations apply when adding database objects to a repository:

- Data is not included when you add cubes or views to a repository.
- Attributes are not included when you add dimensions to a repository.

Git repository integration is also available through the TM1 REST API.

# Add database objects to a Git repository

You can add public database objects to a Git repository from the Databases tree on a modeling workbench. You cannot add private objects to a Git repository.

## About this task

Before you can add database objects to a repository, you must generate an SSH public/private keypair on your computer and add the public key to your repository account.

You can add these objects to a repository:

- cubes (rules and public views)
- dimensions (hierarchies and public sets)
- processes
- chores
- websheets
- control cubes
- control dimensions
- control processes
- files

By default, all associated public views are included when you add a cube to a repository. If you want to exclude views, enable the **Exclude cube views** option on the Settings.

The following limitations apply when adding database objects to a repository:

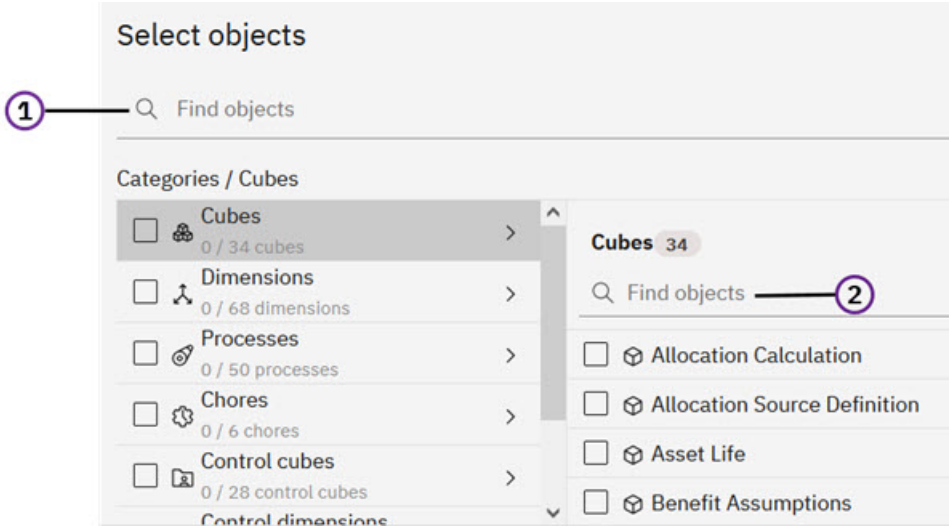
- Data is not included when you add cubes or views to a repository.
- Attributes are not included when you add dimensions to a repository.

## Procedure

1. Open the **Databases** tree on a modeling workbench.

2. Click the **Action** icon ⋮ on the database that contains the objects you want to add to a repository.
3. Click **Add objects to repository**.
4. Enter **Authentication** information. This information is stored only for the duration of the current web session. You can also reset the connection and remove the stored authentication information at any time.
   a) Enter the SSH **URL** for the repository.
   b) Enter your **Public key** for the repository.
   c) Enter your **Private key** for the repository.
   d) Enter the **Passphrase** for your private key. If there is no passphrase associated with your private key, leave the **Passphrase** option empty.
   e) Click **Next**.
5. On the **Select objects** panel, select the objects you want to add to the repository. Your **Settings** determine how dependent objects are handled when you select or unselect an object with dependencies.
   a) To select all objects in a category, click the box next to the category name.
   b) To select individual objects within a category, click the category name, then click the box next to each object you want to add.

      You can also use **Search** to find objects that you want to select. There are two **Search** options on the **Select objects** panel.




      **1**
      Global search of all objects on the database.

      **2**
      Search within the selected object category.

   c) To remove objects from the **Selected objects** list, click **Remove** on the object name.
   d) Click **Next**.
6. Select the repository branch to which you want to add the database objects.
   a) To add to an existing branch, click **Existing branch** and then select the branch from the **Branch name** list.
   b) To add to a new branch, click **New branch**, then enter the new **Branch name** and use the **Create branch based on** option to select the base branch for the new branch.
   c) Enter a message that describes your actions. This is mandatory.
   d) Click **Add**.

### Results

All selected objects are added to the specified repository branch and a brief message indicates the status of the repository update.

While objects are being added to the repository, the database icon on the Databases tree shows an 'updating' overlay.




## Git repository integration settings

Git repository integration settings determine how dependent objects are handled when you select or unselect objects to add to a repository.

To open the settings, click the **Settings** icon on the **Add objects to repository** page.

The **Select actions** options determine what happens when you select an object with dependencies to add to a repository.

**Select dependencies**
: Enable this option to automatically select any dependencies when you select an object. For example, if you select a dimension that is included in two cubes, the two cubes are also selected.

  Clear this option to ignore any dependencies when you select an object.

**Exclude cube views**
: Enable this option to exclude views when you add cubes to the repository.

The **Unselect actions** options determine what happens when you unselect an object with dependencies from the list of selected objects.

**Ask when required**
: Present a prompt to let the user decide how to handle the dependent objects.

**Unselect objects and no action on dependencies**
: Unselect the object, but do nothing to any dependent objects. For example, if you unselect a dimension that is used in a cube that is also selected for addition to the repository, unselect only the dimension.

**Unselect objects and dependencies**
: Unselect the object and also unselect any dependent objects. For example, if you unselect a dimension that is used in a cube that is also selected for addition to the repository, the cube is automatically unselected..

**No action on objects with dependencies**
: Do nothing when an object with dependencies is unselected.

# Update your model with database objects from a Git repository

You can update your database with objects from a Git repository directly from the Databases tree on a modeling workbench.

### About this task

You can't select object categories or individual objects when you update your model with database objects from a Git repository.

### Procedure

1. Open the **Databases** tree on a modeling workbench.
2. Click the **Action** icon on the database that you want to update with objects from a repository.
3. Click **Update model from repository**.

4. Enter **Authentication** information. This information is stored only for the duration of the current web session.
   a) Enter the SSH **URL** for the repository.
   b) Enter your **Public key** for the repository.
   c) Enter your **Private key** for the repository.
   d) Enter the **Passphrase** for your private key. If there is no passphrase associated with your private key, leave the **Passphrase** option empty.
   e) Click **Authenticate**.
5. Select the repository branch that contains the objects you want to use to update your model, then click **Update**.

   An **Update plan** shows you object categories and individual objects that will be updated on the database. You can review the plan, but you cannot modify it. For each object, an action in parentheses indicates how the object will be managed on the database.
6. Click **Execute**.

# Reset a Git repository connection

You can reset the connection between a database and a Git repository from a modeling workbench.

## About this task

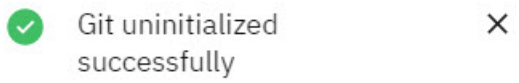
Resetting a connection between a Git repository and a specific database has no impact on any active repository connections on other databases.

## Procedure

1. Open the **Databases** tree on a modeling workbench.
2. Click the **Action** icon ⋮ on the database for which you want to reset the repository connection.
3. Click **Reset connection to repository**.

## Results

The connection to the Git repository is closed and your stored authorization credentials are deleted. A brief message indicates that the Git connection was successfully reset.



# Chapter 7. Administer IBM Planning Analytics Workspace

Administering IBM Planning Analytics Workspace includes managing users, administering databases, migrating assets, and other administrative actions. You access the **Administration** page from the **Administration** tile on the Planning Analytics Workspace home page.

If you are not an administrator, you cannot see the **Administration** tile on the Planning Analytics Workspace home page.

## Download a Welcome Kit from Planning Analytics Administration

When you receive initial notification that a Planning Analytics Welcome Kit is available, you must download the kit from IBM@Box. However, any subsequent Welcome Kits can be downloaded directly from Planning Analytics Administration.

### Before you begin

You must be a Subscription Administrator to download a Welcome Kit from Planning Analytics Administration

If you have multiple tenants (environments), Welcome Kits for all tenants must be downloaded from your primary tenant.

.

### About this task

After you download your initial Welcome Kit from IBM@Box, some changes can result in a new Welcome Kit being required for your tenant. For example, if you add a modeler license or change the password for your shared folder, you'll need to download an updated Welcome Kit.

### Procedure

1. Log in to your primary Planning Analytics Workspace tenant as a Subscription Administrator.
2. Click the **Administration** tile.
3. At the top of the Administration page, click the **Key** icon .
4. On the **Welcome kit settings** page, click the **Welcome Kit** download link.

   If you have multiple tenants, you can pick the tenant for which you want to download the Welcome Kit.

   The Welcome Kit is saved to your Downloads directory with this naming format: *tenant_name* `Welcome Kit.zip`.

   See Contents of the Welcome Kit for a full description of the Welcome Kit.

## Administer users and groups

To administer users in IBM Planning Analytics Workspace, you must be logged in to Planning Analytics Workspace as an **administrator**. If you have the administrator role and meet all of the requirements, you can invite people who are in the same organization (account) as you to become users in Planning Analytics Workspace on cloud or upload users to Planning Analytics Workspace Local.

© Copyright IBM Corp. 2009, 2025

# Administer users on cloud

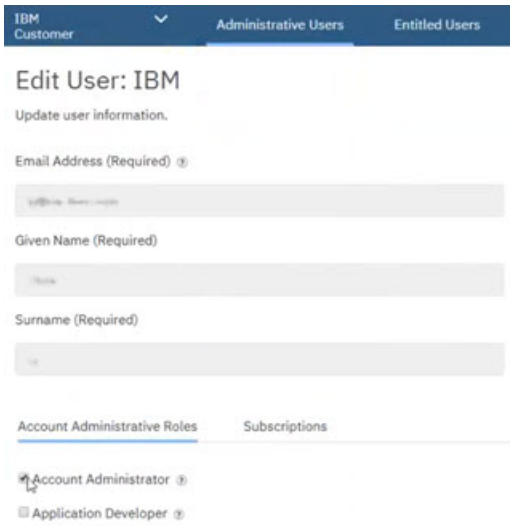
To administer users in IBM Planning Analytics Workspace on cloud, you must understand the intersection between subscriptions, environments, and roles. Your user profile must also have a few more characteristics.

## What is the profile of an administrator who can administer users on cloud?

The user profile of an administrator on cloud shows the full name, user ID, contact email, role, subscription, environments, and groups.

When you are an account administrator for multiple organizations, you must use https://myibm.ibm.com to manage users in any organization other than the first organization to which you were added.

You can verify your role as an account administrator for your organization in the Subscription and Subscriber Management tool.




## What's an organization versus an environment?

An organization has one primary environment and in addition, can have other environments. For example, your primary environment is your production environment and you can also have a non-production, or test, environment. A user can have access to multiple environments. A user can have one role and one subscription in an environment.

**Important:** When a user who is a member of more than one environment accepts an email invitation to a newly created environment, the provisioning of the user in the new environment might not be complete before the user accepts the invitation. Waiting a short time and accepting the invitation again redirects the user to the new environment or to an environment selector with the new environment in the list of available environments.

## What do I need to know about subscriptions?

In Planning Analytics Workspace on cloud, the subscription determines the types of roles that a user can have.

In the following table, a user with the subscription shown in the column heading can have the roles in the row marked with a check mark.

For example, if a user has an IBM Planning Analytics User subscription, then they can be assigned the Consumer or Analyst role.

| Table 13. The intersection between subscriptions and roles | | | |
|---|---|---|---|
| | **Planning Analytics Modeler subscription** | **Planning Analytics User subscription** | **Planning Analytics Explorer subscription** |
| Consumer role | ✔ | ✔ | ✔ |
| Analyst role | ✔ | ✔ | |
| Modeler role | ✔ | | |
| Administrator role | ✔ | | |

### Workflow

Follow these steps to manage a group of new users on cloud:

1. **Invite user** to join Planning Analytics Workspace. The user has a status of *Invited but not activated*. The user receives an invitation by email. If a user accepts the invitation, the user has a status of *Active*. For more information, see "Add and invite a user (cloud only)" on page 877.
2. **Upload users** using a CSV file and then invite them to use Planning Analytics Workspace. In Planning Analytics Workspace on cloud, all users have the Analyst role and the same subscription as you. For more information, see "Upload a file to add or remove multiple users (cloud only)" on page 878.
3. **Create** groups of users in the **Groups** tab. For more information, see "Administer groups" on page 901.
4. Optional: **Upload groups** in the **Groups** tab by dropping a CSV file on to the dialog box or tapping the dialog box to browse for a local CSV file. For more information, see "Administer groups" on page 901.
5. **Export users** to a CSV file. For more information, see "Export users" on page 902.
6. Click **Export groups** in the **Groups** tab to save a CSV file of existing groups to your local file system. This is a great way to create a CSV file to add users to existing groups.
7. Optional: **Manage roles** for several users at one time. This is useful if you upload a group of users but want to assign them different roles. For more information, see "Change a user's role" on page 903.
8. Optional: **Manage subscriptions** for several users at one time. This approach is useful if you upload a large group of users but want to change individual subscriptions. For more information, see "Change a user's subscription (cloud only)" on page 883.
9. Optional: **Manage environments** for a user. For more information, see "Change a user's environment (Cloud only)" on page 904.
10. Optional: **Manage user accounts** to add or delete a user as a subscription administrator. For more information, see "Manage user invitation capability (cloud only)" on page 875.

## Manage user invitation capability (cloud only)

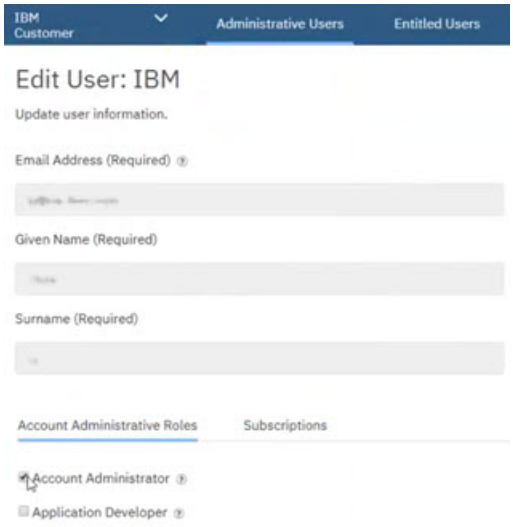
Your organization (account) can have multiple users that are account administrators. Account administrators can invite other users to your organization.

### *Manage user invitation capability in Planning Analytics Workspace*

### Before you begin

When you are an account administrator for multiple organizations, you must use https://myibm.ibm.com to manage users in any organization other than the first organization to which you were added.

You can verify your role as an account administrator for your organization in the Subscription and Subscriber Management tool.




## Procedure

1. On the **Welcome** page, click the **Administration** tile.
2. Click **Users & Groups**.
3. Click **IBM Subscriptions Management**.

   The organization (account) that you belong to is displayed in the upper left of the window.
4. Click **Entitled Users**.

   The list that displays shows all the users that are members of your organization. For each user, their name, email address, subscription, and entitlement status is displayed.
5. In the list, click the name of the user that you want to make an account administrator.
6. In the **Edit User** window, click **Account Administrative Roles**, and select the **Account Administrator** check box.

   **Remember:** The user can be an account administrator only for the first organization that they were added to.
7. Click **Save and Close**.

### *Manage user invitation capability in Planning Analytics Workspace Classic*

## Before you begin

When you are an account administrator for multiple organizations, you must use https://myibm.ibm.com to manage users in any organization other than the first organization to which you were added.

## Procedure

1. On the **Welcome** page, click the **Administration** tile.
2. On the **Users** page, click **Manage user accounts**.

   The organization (account) that you belong to is displayed in the upper left of the window.
3. Click **Entitled Users**.

   The list that displays shows all the users that are members of your organization. For each user, their name, email address, subscription, and entitlement status is displayed.
4. In the list, click the name of the user that you want to make an account administrator.
5. In the **Edit User** window, click **Account Administrative Roles**, and select the **Account Administrator** check box.

**Remember:** The user can be an account administrator only for the first organization that they were added to.

6. Click **Save and Close**.

## Add and invite a user (cloud only)

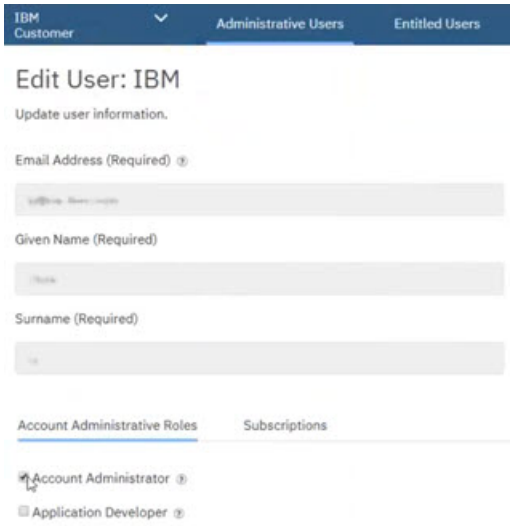
To add and invite users one at a time, follow these instructions.

### *Add and invite a user in Planning Analytics Workspace*

### Before you begin

When you are an account administrator for multiple organizations, you must use https://myibm.ibm.com to manage users in any organization other than the first organization to which you were added.

You can verify your role as an account administrator for your organization in the Subscription and Subscriber Management tool.




### About this task

If you invite a single user at a time, you must invite the user to your primary environment first. After the user accepts the invitation to the primary environment, you can add them to other environments. A user becomes active after they accept the invitation.

### Procedure

1. On the **Welcome** page, click the **Administration** tile.
2. Click **Users & Groups**.
3. Click the **Users** tab.
4. Click **Invite a new user** + and enter their first name, last name, and email address.
5. Select a role.

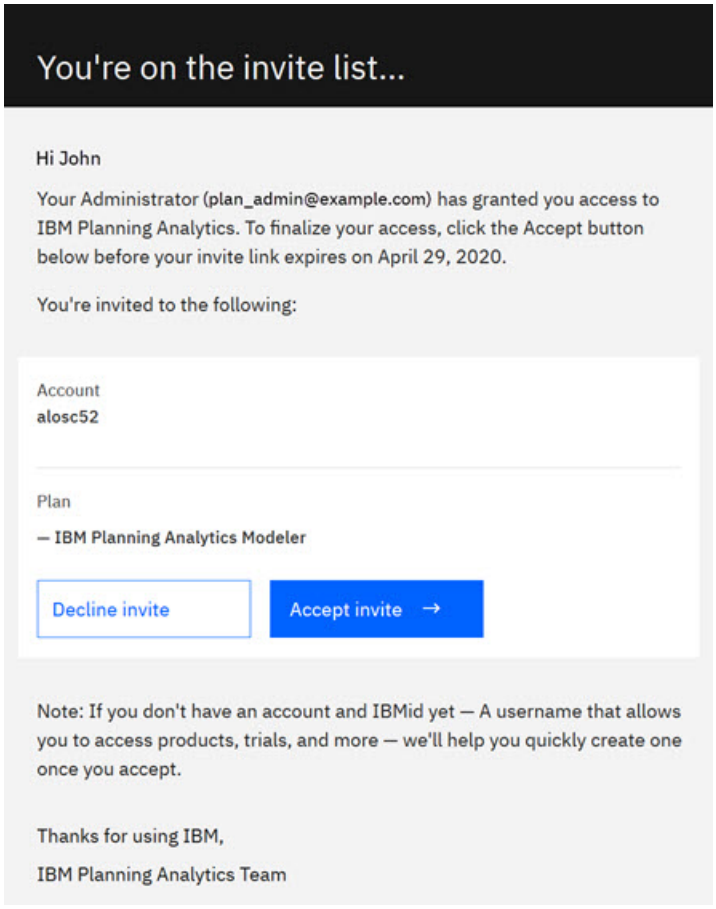
   The Analyst role is selected by default. You can change the role later. For more information about roles, see “User roles” on page 415.
6. Select a subscription for the user.

   **Note:** When inviting a user to Planning Analytics Workspace in Planning Analytics as a Service, there is no option select a subscription.
7. Click **Submit**.

   After a user is invited, their status is **Invited but not activated**.

### Results

When you send an invitation, the user receives an email similar to this. The invitation clearly shows the email address of the administrator who sent the invitation, as well as the account and plan associated with the invitation.




The invitation is valid for 28 days. A user cannot access Planning Analytics Workspace if they do not accept the invitation. If the user declines the invitation or does not accept within 28 days, the invitation is revoked and you must send a new invitation to grant access to the user.

Note that invitation and confirmation emails come from several different IBM systems and email accounts. Tell your users to expect these emails.

After the user accepts the invitation and logs in to Planning Analytics Workspace, their status changes to **Active**.

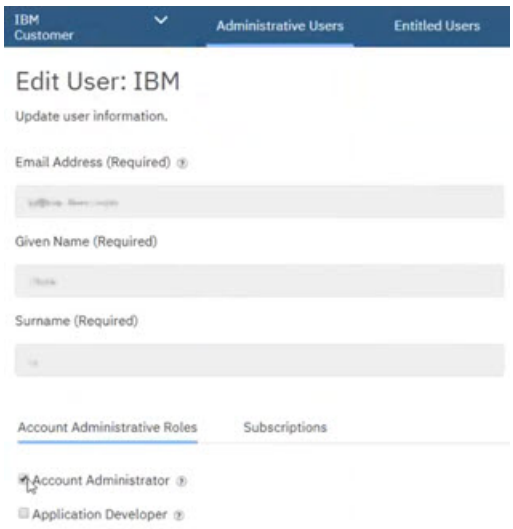
## Upload a file to add or remove multiple users (cloud only)

To add or remove multiple users, upload a file that includes user login IDs and ADD or REMOVE directives. Users that are added receive an invitation to the environment specified in the file.

### Before you begin

When you are an account administrator for multiple organizations, you must use https://myibm.ibm.com to manage users in any organization other than the first organization to which you were added.

You can verify your role as an account administrator for your organization in the Subscription and Subscriber Management tool.




## About this task

- To add and/or remove multiple users in Planning Analytics Workspace on cloud, create a CSV file that contains one line for each user with the following information: Login ID, first name, last name, role, environment, directive.

  The login ID is the email address that is used to sign in to Planning Analytics Workspace. Login ID, first name, and last name are mandatory. All other attributes are optional. If you omit role, it defaults to Analyst. If you omit environment, it defaults to the environment that you are logged in to. If you omit directive, it defaults to ADD.

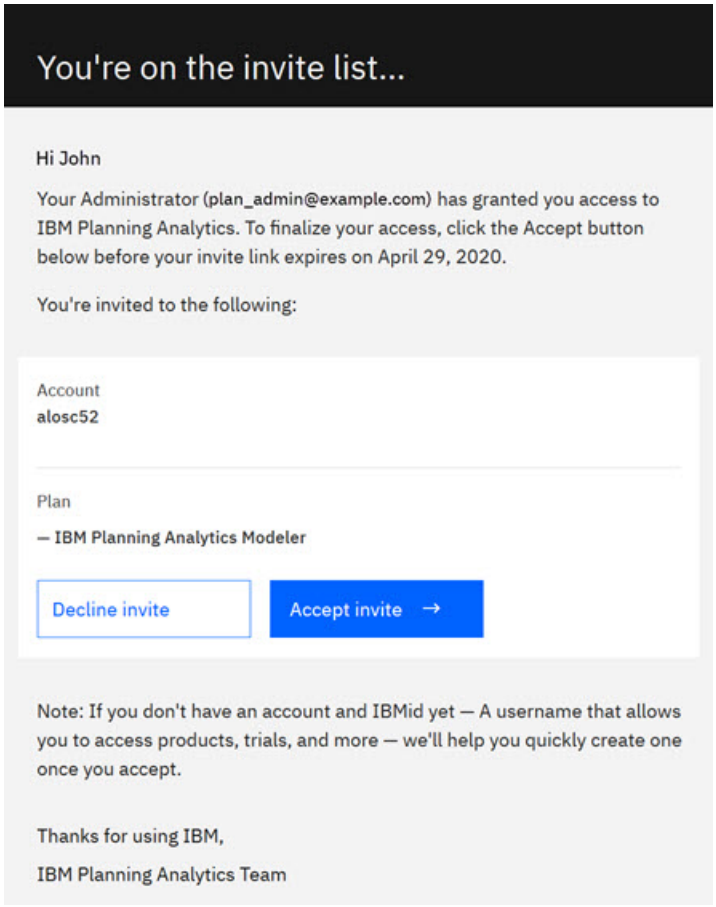
  **Note:** When uploading users from a .csv file, please save the file with the CSV UFT-8 format encoding to ensure that special characters are imported.

| | A | B | C | D | E | F |
|---|---|---|---|---|---|---|
| 1 | **Login ID** | **First Name** | **Last Name** | **Role** | **Environment** | **Directive** |
| 2 | Mary@example.com | Mary | Marks | Administrator | dev environment | REMOVE |
| 3 | Nolan@example.com | Nolan | Michaels | Analyst | prototype environment | ADD |
| 4 | Robert@example.com | Robert | Jordan | Modeler | dev environment | ADD |
| 5 | Robert@example.com | Robert | Jordan | Modeler | prototype environment | ADD |
| 6 | Lauri@example.com | Lauri | Sterling | Consumer | prototype environment | REMOVE |

- If you invite multiple users at a time, the number of users that you can invite is limited to the number of actual available Planning Analytics Workspace subscriptions you purchased, exclusive of any overage allowances. For example, if you have seven subscriptions available with unlimited overage, you can invite up to seven users. If you invite a number of users greater than the number of actual available subscriptions, the invitation fails.
- You must invite users to your primary environment first and they must be active in the primary environment before you can upload them to any secondary environments.
- You cannot assign a user to an environment that they already belong to, nor to an environment that does not exist.
- You can add users to multiple secondary environments by using a single CSV file.
- A user can have different roles in different environments. For more information about roles, see “User roles” on page 415.
- You can use file upload to remove users only from a non-primary environment. To remove a user from the primary environment, see “Remove a user (cloud only)” on page 881. You must remove a user from all non-primary environments before you remove them from the primary environment.
- You can use a file to add or remove up to 100 users at a time.

## Results

When you send an invitation, the user receives an email similar to this. The invitation clearly shows the email address of the administrator who sent the invitation, as well as the account and plan associated with the invitation.




The invitation is valid for 28 days. A user cannot access Planning Analytics Workspace if they do not accept the invitation. If the user declines the invitation or does not accept within 28 days, the invitation is revoked and you must send a new invitation to grant access to the user.

Note that invitation and confirmation emails come from several different IBM systems and email accounts. Tell your users to expect these emails.

After the user accepts the invitation and logs in to Planning Analytics Workspace, their status changes to **Active**.

### *Upload a file to add or remove multiple users in Planning Analytics Workspace*

## Procedure

1. On the **Welcome** page, click the **Administration** tile.
2. Click **Users & Groups**.
3. Click the **Users** tab.
4. Click **Upload users** .
5. Drag the CSV file to the **Drop your .csv file here** region , or browse to the file location and import the file.

   The file is uploaded and you see a success message.
6. Click **OK**.

The users that you added with the ADD directive in the CSV file appear on the **Users** tab with a status of **Not invited yet**.

7. To invite users that you uploaded with the ADD directive, select the users with a status of **Not invited yet**.
   a) Click **Send invitation** in the **Applicable actions** list.
   b) Select a subscription for the invited users.
   c) Click **Submit**.
8. To delete users that you uploaded with the REMOVE directive, select the users.
   a) Click **Delete users** in the **Applicable actions** list.
   b) Click **OK**.

## Remove a user (cloud only)

As an administrator, you can remove users from IBM Planning Analytics Workspace on cloud.

**Note:** Removing a user's subscription to the Planning Analytics on Cloud service also renames the user's personal folder to **Unknown**. All the workspace assets of the removed user remain in the renamed folder. As an administrator, you can see but cannot open assets in the personal folder of a user that has been removed. You must move the assets from the removed user's personal folder and place them in your own personal folder or the Shared folder to open them.

### Procedure

1. If you want to completely remove a user from all subscriptions to IBM products, see "Remove a user from all subscriptions" on page 882.
2. If the user has access to multiple subscriptions from other IBM products that they want to keep, see "Remove a user from Planning Analytics Workspace only " on page 882.

### *Remove users in Planning Analytics Workspace*

Planning Analytics on Cloud administrators can remove multiple users at once in Planning Analytics Workspace **Administration**.

After one or more users are removed from the primary environment by the administrator, the following changes are applied automatically:

- The users are removed from all secondary environments, from IBM Subscriptions Management, and from group memberships.
- The users' personal folders are renamed to **Unknown**, and each user's workspace assets remain in this folder.

  As an administrator, you can see but cannot open assets in the personal folders of the removed users. You must move the assets to your own personal folder, or to the **Shared** folder, to open them. Users' shared assets remain in the shared folder.

### Procedure

1. Log on to Planning Analytics as a member of the Administrator role.
2. In the Welcome page, select **Administration**.
3. Click **Users & Groups**.
4. On the **Users** tab, select one or multiple users that you want to remove.
5. In the toolbar, click the delete icon  .

   The **Delete users** dialog box with the selected user name or names is displayed.
6. Click **Yes** to remove the selected users.

   If the names are successfully removed, a confirmation message is displayed.

### *Remove a user from all subscriptions*

You can remove a user from all IBM subscriptions.

## Before you begin

- You must be a member of the same organization as the users that you are removing.
- You must be a subscription administrator.

## Results

- As of Planning Analytics Workspace 2.0.54, the user is automatically removed from all environments within the organization. You no longer need to remove the user from secondary environments before removing the user from the primary environment.
- Removing a user's subscription to the Planning Analytics on Cloud service also renames the user's personal folder to **Unknown**. All the workspace assets of the removed user remain in the renamed folder. As an administrator, you can see but cannot open assets in the personal folder of a user that has been removed. You must move the assets from the removed user's personal folder and place them in your own personal folder or the Shared folder to open them.
- The user will not have access to Planning Analytics Workspace and will no longer consume a Planning Analytics subscription.
- The user is no longer a member of the organization and can be invited to be a member of another organization.
- The user will no longer have access to any other IBM subscriptions that they might be entitled to with this subscription management service.

*Remove a user from all subscriptions in Planning Analytics Workspace*

## Procedure

1. On the **Welcome** page, click the **Administration** tile.
2. Click **Users and Groups**.
3. Click **Users**.
4. Click **IBM Subscriptions Manager**.

   The organization (account) that you belong to opens in the current window.
5. Click **Entitled Users**.

   The list shows all the users that are members of your organization. For each user, you can see their name, email address, organization, subscription, and entitlement status.
6. Click adjacent to the user that you want to remove.
7. Click **Remove Access**.
8. Click **OK** to confirm.

   Allow several minutes for the **Users** tab to update.

### *Remove a user from Planning Analytics Workspace only*

You can remove a user from only the Planning Analytics on Cloud service if a user has access to multiple IBM subscriptions.

## Before you begin

- You must be a member of the same organization as the users that you are removing.
- You must be a subscription administrator.

### Results

- As of Planning Analytics Workspace 2.0.54, the user is automatically removed from all environments within the organization. You no longer need to remove the user from secondary environments before removing the user from the primary environment.
- Removing a user's subscription to the Planning Analytics on Cloud service also renames the user's personal folder to **Unknown**. All the workspace assets of the removed user remain in the renamed folder. As an administrator, you can see but cannot open assets in the personal folder of a user that has been removed. You must move the assets from the removed user's personal folder and place them in your own personal folder or the Shared folder to open them.
- The user will not have access to Planning Analytics Workspace and will no longer consume a Planning Analytics subscription.

*Remove a user from Planning Analytics Workspace only in Planning Analytics Workspace*

### Procedure

1. On the **Welcome** page, click the **Administration** tile.
2. Click **Users & Groups** tab.
3. Click the Users tab.
4. On the **Users** page, click **IBM Subscriptions Management**.

   The organization (account) that you belong to opens in the upper left corner of the window.
5. Click **Entitled Users**.

   The list shows all the users that are members of your organization. For each user, you can see their name, email address, organization, subscription, and entitlement status.
6. Click the name of the user that you want to remove.
7. In the **Edit User** window, click **Subscriptions**, and clear the **Planning Analytics Subscription** (Modeler, User, or Explorer) check box.
8. Click **Save and Close**.

   Allow several minutes for the **Users** tab to update.

## Change a user's subscription (cloud only)

You can change the subscription for one or more users at a time. Subscriptions apply only to IBM Planning Analytics Workspace on Cloud. Subscriptions are not applicable to Planning Analytics Local or Planning Analytics as a Service.

### Before you begin

When you are an account administrator for multiple organizations, you must use https://myibm.ibm.com to manage users in any organization other than the first organization to which you were added.

### About this task

Your IBM Planning Analytics license determines the subscriptions that are available for you to assign to a user. For example, if your organization has a standard subscription, then you can assign one of the following subscriptions to a user:

- IBM Planning Analytics Explorer
- IBM Planning Analytics User
- IBM Planning Analytics Modeler

### *Change a user's subscription in Planning Analytics Workspace*

#### Procedure

1. On the **Welcome** page, the **Administration** tile.
2. Click **Users & Groups**.
3. Click **IBM Subscriptions Management**.
4. Click the user for whom you want to change subscriptions.
5. On the **Edit user** window, click the **Subscriptions** tab.
6. Select the subscriptions for the user, then click **Save and close**.

## Manage user capabilities in Planning Analytics Administration (Cloud only)

Administrators can manage some capabilities for individual users in Planning Analytics Administration. The specific capabilities that can be managed will expand over time.

#### About this task

Initially, only a subscription administrator can manage capabilities and only in a primary environment. The subscription administrator can enable the ability to manage capabilities for other administrators, who can then manage capabilities for other individual users. The subscription administrator can also assign the ability for other administrators to manage capabilities on a non-primary environment.

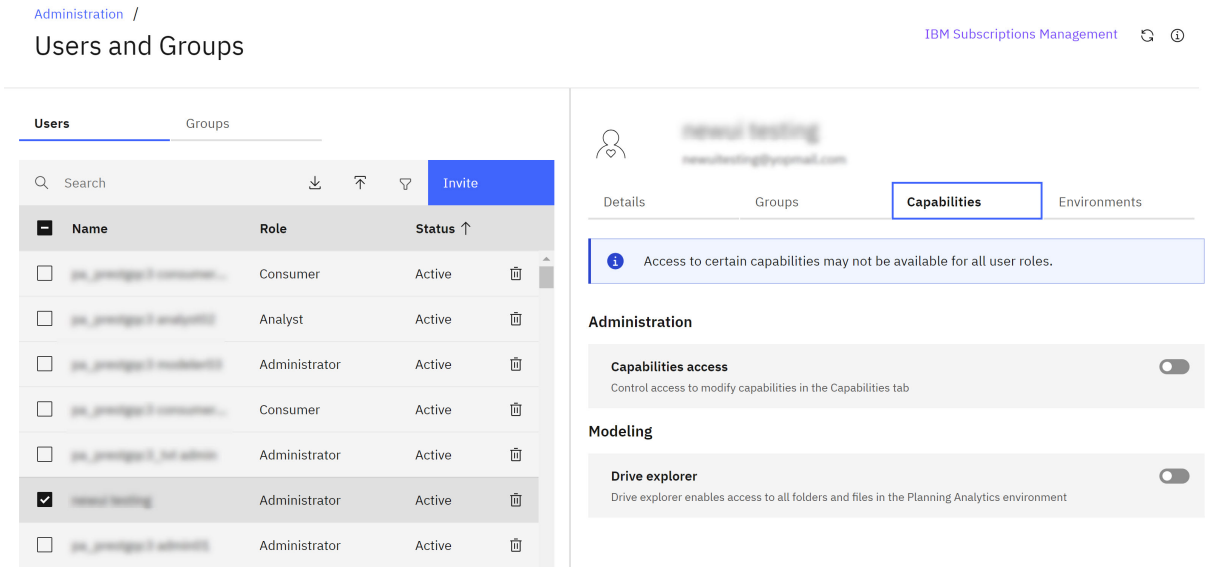
### *Manage user capabilities*

#### About this task

The procedure for managing user capabilities is the same for primary environments and non-primary environments, but to allow capability management in a on-primary environment , a subscription manager must first assign an administrator the ability to manage user capabilities in a non-primary environment.

#### Procedure

1. Open Planning Analytics Administration as a subscription administrator or as an administrator who has been given the ability to manage capabilities.
2. Click the **Users and groups** tile.
3. Click the **Users** tab.
4. Select the user for whom you want to manage capabilities.
5. Click the **Capabilities** tab.




The tab shows the capabilities that can be managed, grouped by functional area.

**Important:** The **Capabilities access** capability grants other administrators the ability to manage capabilities. Be deliberate when enabling this capability, as any administer with **Capabilities access** enabled can in turn enable it for other administrators.

6. Click the toggle to enable or disable a capability for the selected user.

   You can enable any capability for a user, regardless of the user's role. However, a user's ability to access and use a specific capability might depend on the user's current role. You can enable capabilities only for users that are members of that environment.

   For example, you can enable the **Drive explorer** capability for a user with the Analyst role. The **Drive explorer** is available only on a modeling workbench, so an Analyst would never be able to access the capability. If, however, the user's role is later changed to Modeler, the user could then access the enabled **Drive explorer** capability.

## *Assign an administrator the ability to manage user capabilities in a non-primary environment*

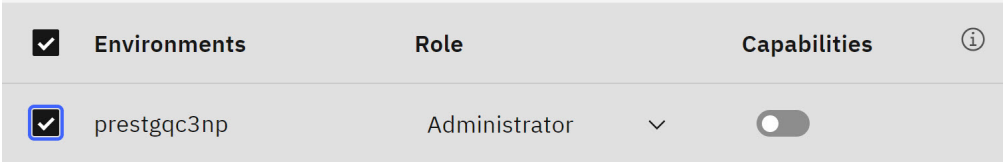
### About this task

For a non-primary environment, a subscription administrator must first give an administrator in the non-primary environment the ability to manage **Capabilities access**. In most cases, you'll want to identify a single administrator in a non-primary environment to manage capabilities and enable **Capabilities access** for the administrator. That administrator can then manage capabilities as described in "Manage user capabilities " on page 884.

### Procedure

1. Open Planning Analytics Administration as a subscription administrator.
2. Click the **Users and groups** tile.
3. Click the **Users** tab.
4. Select the user you want to manage capabilities in a non-primary environment.
5. Click the **Environments** tab.
6. Click **Manage**.

   The **Manage environments** page displays all the non-primary environments that are available. If the selected user is already a member of a non-primary environment, the checkbox next to the non-primary environment name is selected.




7. If the user is not currently a member of the non-primary environment, fill the checkbox next to the non-primary environment name.
8. If the user is not an administrator on the non-primary environment, click the **Role** menu and select **Administrator**.

   **Note:** You can assign the ability to manage user capabilities in a non-primary environment only to a user with the Administrator role. The **Capabilities** toggle on the **Manage environments** page remains disabled for all other user roles.
9. Enable the **Capabilities** toggle.
10. Click **Save**.

    The user is now an administrator on the non-primary environment with the ability to manage user capabilities.

## View your subscription consumption and possible overage (cloud only)

As an administrator, you can view your total Planning Analytics subscription entitlement and determine how many subscriptions of your total entitlement are available. You must be a subscription administrator to view subscription consumption.

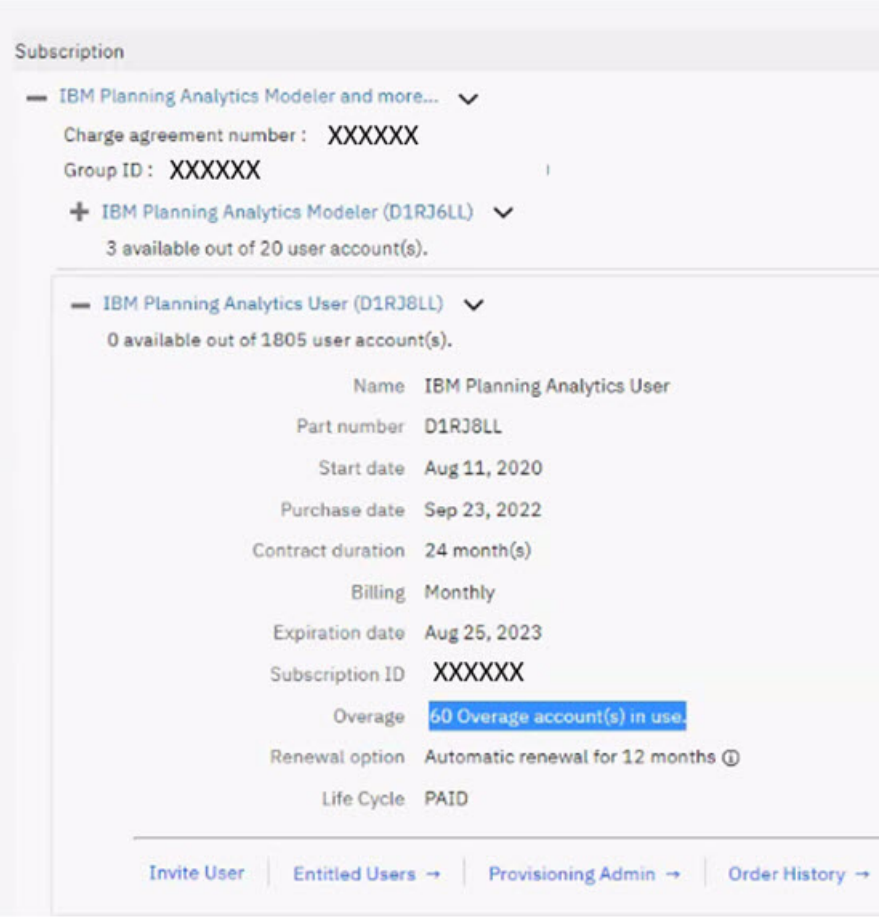
### *View your subscription consumption in Planning Analytics Workspace*

### Procedure

1. On the Planning Analytics Workspace home page, click the **Administration** tile.
2. Click **Users & Groups**.
3. Click **Subscription Management.**

   On the **IBM Subscription Management** page, the organization (account) that you belong to is displayed on the window.
4. Click **Subscriptions**.
5. Click any of the category headings (IBM Planning Analytics Modeler, IBM Planning Analytics User, IBM Planning Analytics Explorer) to view details and determine how many user accounts of your total entitlement are available.

   Overage accounts indicate that your organization is using more than it is entitled to. These accounts are subject to overage fees according to your contractual agreement.




In this example, 1805 IBM Planning Analytics User licenses were purchased and there is a current overage of 60 accounts in use. This means a total of 1865 user accounts are currently active within this environment, 60 more than the purchased entitlements.

### Understand who is a default member of the ADMIN security group in TM1

All users with the Administrator role in Planning Analytics Workspace are included in the Cognos group with `CAMID("pans:g:Subscription Administrators")`.

When a TM1 database is created, a new CAM associated group entry is created in the control cube `}ClientCAMAssociatedGroups`, which maps the Cognos group `CAMID("pans:g:Subscription Administrators")` to the generic TM1 security group `ADMIN`. When a user is assigned the Administrator role in Planning Analytics Workspace, they are implicitly added to the `CAMID("pans:g:Subscription Administrators")` group, and are automatically added to the ADMIN security group in TM1.

All TM1 security groups, including the ADMIN group, are described in the TM1 Operations Guide.

## Manage users and groups on a Planning Analytics database (Cloud only)

Use the **Users and groups** editor to manage users and groups on a Planning Analytics database. You can also use the editor to import users and groups from Planning Analytics Workspace into a Planning Analytics database. You must be an administrator or modeler to manage users and groups on the database.

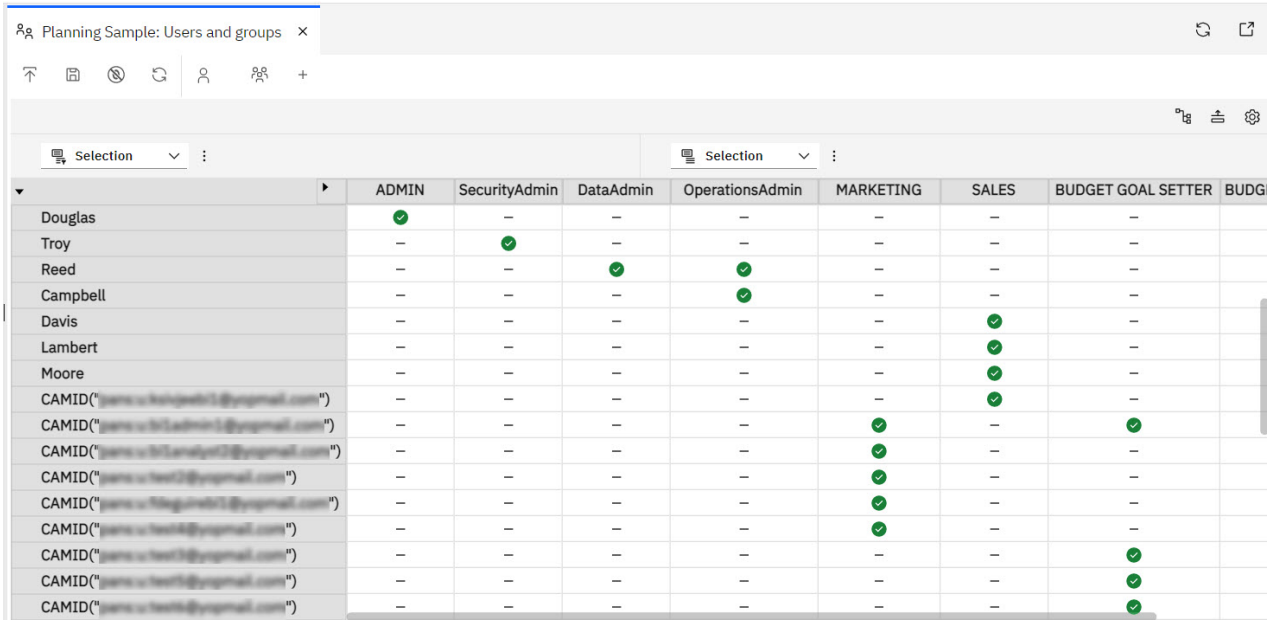
To open the **Users and groups** editor on a modeling workbench, right-click a database on the **Databases** tree and then click **Manage users and groups**.

# Manage existing users and groups on your Planning Analytics database (Cloud only)

As an administrator or modeler, you can manage existing users and groups on your Planning Analytics database and create new groups on the database.

## About this task

The **Users and groups** editor displays your current database users and groups in an enhanced cube view.




Use the standard view options to manage sets and to select multiple row or column headers in the **Users and groups** editor.

## Procedure

To open the **Users and groups** editor on a modeling workbench, right-click a database on the **Databases** tree and then click **Manage users and groups**.

a) To assign or remove a user from a group, right-click at the intersection of a user and group name, then click **Assign user(s) to group** or **Remove user(s) from group(s)**.

b) To assign all users to a group, right-click the group name, then click **Assign group(s)**.

c) To remove all users from a group, right-click the group name, then click **Unassign group(s)**.

d) To assign a user to all groups, right-click the user name, then click **Assign user(s) to group**.

e) To remove a user from all groups, right-click the user name, then click **Remove user(s) from group(s)**.

f) To remove a group from the database, right-click the group name, then click **Remove group(s)**.

g) To remove a user from the database, right-click the user name, then click **Remove user(s)**.

h) To synchronize a group in the database with Planning Analytics Workspace, right-click the group name, then click **Synchronize group(s)**.

   Settings of the group in the database now match those of the same group that exists in Planning Analytics Workspace.

   Synchronization does not remove any new users that were added to the database. For example, if you add a user from Planning Analytics Workspace to the database and assign it to group Modeler. When you synchronize the group, if that user does not exist in the Modeler group in Planning Analytics Workspace, the user will be unassigned from the Modeler group in the database. However, the user will not be removed from the database.

## Add a user group to a Planning Analytics database (Cloud only)

You can add a new user group to a Planning Analytics database from the modeling workbench.

### Procedure

1. To open the **Users and groups** editor on a modeling workbench, right-click a database on the **Databases** tree and then click **Manage users and groups**.
2. To create a new group on the database, click + .

### *Rename a group (Cloud only)*

Administrators can rename Planning Analytics user groups in Planning Analytics Administration.

### Procedure

1. Log in to Planning Analytics Workspace as a member with the Administrator role.
2. Click the **Administration** tile.
3. Click the **Users and Groups** tile.
4. Click the **Groups** tab.
5. Click the group that you want to rename.
6. Click Edit on the group details pane.
7. Enter a new name for the group, and optionally, a description of the group.
8. Click **Save** .

### What to do next

Any existing content, such as process functions, that references the group name must be updated to refer to the new group name.

## Import users and groups from Planning Analytics Workspace into a Planning Analytics database (Cloud only)

After users have been invited to Planning Analytics Workspace, it's a common task to add the users to one or more of the Planning Analytics databases on your environment.

### About this task

You must be an administrator to import users and groups into a database. Modelers cannot perform this action.

### Procedure

1. To open the **Users and groups** editor on a modeling workbench, right-click a database on the **Databases** tree and then click **Manage users and groups**.
2. To add users that exist in Planning Analytics Workspace to the Planning Analytics database, click . Select the users, then click **Add**.

   The users are added to the database, but not assigned to any groups. For information on assigning users to groups, see “Manage existing users and groups on your Planning Analytics database (Cloud only)” on page 888.
3. To add groups that exist in Planning Analytics Workspace to the Planning Analytics database, click . Select the groups, then click **Add**.

   The groups are added to the database. In addition, any users within the groups are also added to the database and assigned to the appropriate groups.

# Administer local users

To administer users in IBM Planning Analytics Workspace Local, you must have the **Administrator** role. Your user profile shows your full name, user ID, role, and groups that you belong to.

## Workflow

Follow these steps to manage a group of users locally:

1. **Upload users** by dropping a CSV file on to the dialog box or tapping the dialog box to browse for a local CSV file. .
2. **Create** groups of users in the **Groups** tab. For more information, see “Administer groups” on page 901.
3. Optional: **Upload groups** in the **Groups** tab by dropping a CSV file on to the dialog box or tapping the dialog box to browse for a local CSV file. For more information, see “Administer groups” on page 901.
4. **Export users** to a CSV file. For more information, see “Export users” on page 902.
5. Click **Export groups** in the **Groups** tab to save a CSV file of existing groups to your local file system. This is a great way to create a CSV file to add users to existing groups.
6. Optional: **Manage roles** for several users at one time. This is useful if you upload a group of users but want to assign them different roles. For more information, see “Change a user's role” on page 903.
7. Optional: **Activate or deactivate a user**. For more information, see “Activate or deactivate a user (Local only) ” on page 895.
8. Optional: **Delete users**.
9. Optional: **Delete multiple users** by using a CSV file. For more information, see “Delete multiple users (Local only)” on page 896.

# Import users to Planning Analytics Workspace from the TM1 authentication database (Local only)

Administrators can import users from the TM1 database that is used for authentication into Planning Analytics Workspace.

You need to have a TM1 authentication database to use this feature. The database that is used for authentication is identified by the icon. The icon is displayed only on Planning Analytics Administration Local and only when Planning Analytics Workspace is configured to authenticate against a specific Planning Analytics database.

**Important:** For the authentication database to be correctly identified in Planning Analytics Administration, the TM1LoginServerURL configuration parameter for Planning Analytics Workspace Local must be specified with an IP address and port. The IP address must match the address that is specified by the IPAddressV4 parameter in your `tm1s.cfg` file. The port number must match the port that is specified by the HttpPortNumber parameter in your `tm1s.cfg` file.

```
TM1LoginServerURL=https://
IPAddressV4_from_tm1s.cfg:HttpPortNumber_from_tm1s.cfg
```

If you have multiple tenants (environments), each tenant must be configured so that the authentication database is available from each tenant.

To import users from the TM1 authentication database, follow these steps:

1. In Planning Analytics Administration, go to **Users and Groups**.
2. On the **Users** tab, select the role and user that you want to import from the TM1 authentication database.
3. Click **Import**.

**Note:** You can import only users, not user groups.

## Add multiple users in Planning Analytics Workspace(Local only)

To add multiple users to IBM Planning Analytics Workspace Local, you upload a list of users with a `.csv` file.

### Before you begin

To administer users in IBM Planning Analytics Workspace Local, you must have the **Administrator** role.

### About this task

Settings for the ENABLE_USER_IMPORT configuration parameter affect when a user can log in to Planning Analytics Workspace Local.

For more information, see Planning Analytics Workspace configuration parameters (https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_inst.2.0.0.doc/c_paw_config_file.html).

### *Add users in Planning Analytics Workspace*

### Procedure

1. Create a `.csv` file that contains the users that you want to upload.

   **Tip:** If you use Microsoft Excel to create the `.csv` file, and the file includes user names with special characters (such as diacritics), you must explicitly set the character encoding to `65001: Unicode (UTF-8)`. For more information, see How to open UTF-8 CSV file in Excel without mis-conversion of characters.

   The `.csv` file can have a header row. If you use a header row, you can order the user information however you want as long as the names of the columns match the supported names. For example:

   ```
   Login ID, First Name, Last Name, Email, Status
   Planning Analytics/Mary Smith,Mary,Smith,msmith@ca.ibm.com, Active
   Planning Analytics/Robert Smith,Robert,Smith,roberts@ca.ibm.com, Inactive
   Planning Analytics/Kevin Alexander,Kevin,Alexander,kalex@ca.ibm.com, Active
   ```

   The `.csv` file must contain one line for each user with the following information: Login ID, First Name, Last Name, Role, Email, Status.

   Login ID, First Name, Last Name, and Status are required. Role and Email are optional.

   If you omit Role, it defaults to Analyst. For information about roles, see “User roles” on page 415.

   If you omit Email, it defaults to Login ID. If the Login ID is not a valid email address, the user will be added with a state of `Not invited yet` and cannot log in to Planning Analytics Workspace Local.

   Status can be `Active` or `Inavtive`.A user that has a status of Inactive can't log in to Planning Analytics Workspace Local until you activate them.

   Text values in the `.csv` file such as role and status must be in English.

   If the `.csv` file does not have a header row, use commas to ensure that the information is in this order: Login ID, First Name, Last Name, Role, Email, Status. For example:

   ```
   Planning Analytics/Mary Smith,Mary,Smith,,msmith@ca.ibm.com,Active
   Planning Analytics/Robert Smith,Robert,Smith,,roberts@ca.ibm.com,Inactive
   Planning Analytics/Kevin Alexander,Kevin,Alexander,Administrator,kalex@ca.ibm.com,Active
   ```

   The first two entries do not specify a Role, so there is an empty string following Last Name.
2. On the **Welcome** page, click the **Administration** tile.

   If you don't see the **Administer** option, then you are not logged in as administrator and you cannot add users in Planning Analytics Workspace Local.
3. Click the **Users & Groups** tab.

4. Click the **Users** tab.
5. Click **Upload users from a .csv file**  .
6. Drag the `.csv` file onto the **Drop your .csv file here** region  , or browse to the file location and import the file.

   The file is uploaded and you see a success message.
7. Click **OK**.

   The users appear under the **Users** tab with the status that you specified in your `.csv` file.

## What to do next

- "Administer groups" on page 901.
- "Change a user's role" on page 903.
- "Activate or deactivate a user (Local only) " on page 895.

### *Add users in Planning Analytics Workspace Classic*

## Procedure

1. Create a CSV file that contains the users that you want to upload.

   **Tip:** If you use Microsoft Excel to create the `.csv` file, and the file includes user names with special characters (such as diacritics), you must explicitly set the character encoding to `65001: Unicode (UTF-8)`. For more information, see How to open UTF-8 CSV file in Excel without mis-conversion of characters.

   The CSV file can have a header row. If you use a header row, you can order the user information however you want as long as the names of the columns match the supported names. For example:

   ```
   Login ID, First Name, Last Name, Email, Status
   Planning Analytics/Mary Smith,Mary,Smith,msmith@ca.ibm.com, Active
   Planning Analytics/Robert Smith,Robert,Smith,roberts@ca.ibm.com, Inactive
   Planning Analytics/Kevin Alexander,Kevin,Alexander,kalex@ca.ibm.com, Active
   ```

   The CSV file must contain one line for each user with the following information: Login ID, First Name, Last Name, Role, Email, Status.

   Login ID, First Name, Last Name, and Status are required. Role and Email are optional.

   If you omit Role, it defaults to Analyst. For information about roles, see "User roles" on page 415.

   If you omit Email, it defaults to Login ID. If the Login ID is not a valid email address, the user will be added with a state of `Not invited yet` and cannot log in to Planning Analytics Workspace Local.

   Status can be `Active` or `Inactive`. A user that has a status of Inactive can't log in to Planning Analytics Workspace until you activate them.

   Text values in the CSV file such as role and status must be in English.

   If the CSV file does not have a header row, use commas to ensure that the information is in this order: Login ID, First Name, Last Name, Role, Email, Status. For example:

   ```
   Planning Analytics/Mary Smith,Mary,Smith,,msmith@ca.ibm.com,Active
   Planning Analytics/Robert Smith,Robert,Smith,,roberts@ca.ibm.com,Inactive
   Planning Analytics/Kevin Alexander,Kevin,Alexander,Administrator,kalex@ca.ibm.com,Active
   ```

   The first two entries do not specify a Role, so there is an empty string following Last Name.
2. On the **Welcome** page, click your user name, and then click **Administer**.

   If you don't see the **Administer** option, then you are not logged in as administrator and you cannot add users in Planning Analytics Workspace Local.

3. If you are not already there, click the **Users** tab.
4. Click **Upload users**.
5. Drag the CSV file onto the **.csv** icon  , or browse to the file location and import the file.
   The file is uploaded and you see a success message.
6. Click **OK**.

   The users appear under the **Users** tab with the status that you specified in your CSV file.

## Add an individual user in Planning Analytics Workspace (Local only)

You can add an individual user in directly from the **Users and Groups** tile in Planning Analytics Administration Local.

### About this task

You can continue to upload a `.csv` file to add multiple users to Planning Analytics Workspace Local.

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace home page.
2. Click the **Users and Groups** tile.
3. Click the **Users** tab.
4. Click **Add**.
5. Enter the required information for the new user, then click **Add**.

## Promote a user to administrator (Local only)

If your IBM Planning Analytics Workspace administrator is no longer available, you can promote a non-admin user to the administrator role by using the command line. If you're upgrading to Planning Analytics Workspace 2.0.92 or a higher version, you can also use the command line to assign Environments admin privileges to a user.

This approach is the only way to promote a user to the administrator role without logging in to Planning Analytics Workspace with an existing administrator role account. This is also the only way to provide the Environments administration privilege if there is no administrator role user with that privilege.

**Note:** If you are using Planning Analytics Workspace Distributed, replace all instances of `wa-proxy` in the following commands with the `containerID`. You can find the `containerID` by using the `docker ps` command.

1. Run the following command to see a list of users in this environment and their most recent logon. While this command returns some of the same information as the command in Step 2, it includes most recent logon, which might be useful in identifying the user that you want to promote to administrator.
   - On Windows:

     ```
     docker exec -it wa-proxy powershell -command "C:\wa-proxy\tools\runTool.ps1"
     findLoggedInAtleastOnceUsers
     ```

   - On Red Hat Enterprise Linux® 7:

     ```
     docker exec -it wa-proxy bash -C /wa-proxy/tools/runTool.sh findLoggedInAtleastOnceUsers
     ```

   - On Red Hat Enterprise Linux® 8 and 9:

     ```
     sudo podman exec -it wa-proxy bash -C /wa-proxy/tools/runTool.sh
     findLoggedInAtleastOnceUsers
     ```

2. Run the following command to see a list of users in this environment, including their login ID, display name, user ID, and role.

- On Windows:

```
docker exec -it wa-proxy powershell -command "C:\wa-proxy\tools\runTool.ps1"
changeUserRoleToAdmin
```

- On Red Hat Enterprise Linux® 7:

```
docker exec -it wa-proxy bash -C /wa-proxy/tools/runTool.sh changeUserRoleToAdmin
```

- On Red Hat Enterprise Linux® 8 and 9:

```
sudo podman exec -it wa-proxy bash -C /wa-proxy/tools/runTool.sh changeUserRoleToAdmin
```

3. Identify the user ID for the user that you want to promote to the administrator role and run the following command with the appropriate `userID`:
   - On Windows:

```
docker exec -it wa-proxy powershell -command  "C:\wa-proxy\tools\runTool.ps1"
changeUserRoleToAdmin <userId>
```

   - On Red Hat Enterprise Linux® 7:

```
docker exec -it wa-proxy bash -C  /wa-proxy/tools/runTool.sh changeUserRoleToAdmin
      <userId>
```

   - On Red Hat Enterprise Linux® 8 and 9:

```
sudo podman exec -it wa-proxy bash -C  /wa-proxy/tools/runTool.sh changeUserRoleToAdmin
      <userId>
```

## Assigning Environments privileges to a user (Local only)

If you are upgrading to Planning Analytics Workspace 2.0.92 and want to create and manage multiple environments in Planning Analytics Administration Local, you must assign Environments admin privileges to at least one user with the administrator role.

**Note:** If you are a new Planning Analytics Workspace user, installing 2.0.90 or a higher version for the first time, the first user with the administrator role automatically gets Environments admin privileges.

You can promote a non-admin user to an administrator role and assign Environments admin privileges to them in one go with the following command:

- On Windows:

```
docker exec -it wa-proxy powershell -command  "C:\wa-proxy\tools\runTool.ps1"
changeUserRoleToAdmin <userId>
```

- On Red Hat Enterprise Linux® 7:

```
docker exec -it wa-proxy bash -C  /wa-proxy/tools/runTool.sh changeUserRoleToAdmin
      <userId>
```

- On Red Hat Enterprise Linux® 8 and 9:

```
sudo podman exec -it wa-proxy bash -C  /wa-proxy/tools/runTool.sh changeUserRoleToAdmin
      <userId>
```

To assign an existing administrator with Environments admin privileges, run the following command:

- On Windows:

```
docker exec -it wa-proxy powershell -command  "C:\wa-proxy\tools\runTool.ps1"
changeUserRoleToAdmin <userId>
```

- On Red Hat Enterprise Linux® 7:

```
docker exec -it wa-proxy bash -C  /wa-proxy/tools/runTool.sh changeUserRoleToAdmin
      <userId>
```

- On Red Hat Enterprise Linux® 8 and 9:

```
sudo podman exec -it wa-proxy bash -C  /wa-proxy/tools/runTool.sh changeUserRoleToAdmin
      <userId>
```

If you want to remove a user from Environments admin privileges, run the command to see a list of users in the environment and set their `environments_admin_privileges` property to `false`.

## Activate or deactivate a user (Local only)

Users in IBM Planning Analytics Workspace Local can be activated or deactivated.

### Before you begin

To administer users in IBM Planning Analytics Workspace Local, you must have the **Administrator** role.

### Procedure

1. Set the ENABLE_USER_IMPORT configuration parameter to TRUE.

   **Note:** When you change the paw configuration file, you must run `./scripts/paw` for your operating system or click **Start** in the Planning Analytics Workspace administration tool for your changes to take effect. Only services that are affected by the configuration change will restart.

   For more information, see Planning Analytics Workspace configuration parameters (https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_inst.2.0.0.doc/c_paw_config_file.html).
2. On the **Welcome** page, click your user name, and then click **Administer**.

   **Note:** If you don't see the **Administer** option, then you are not logged in as administrator and you cannot activate or deactivate users.
3. To invite users that you uploaded with the ADD directive, select the users with a status of **Not invited yet**. Click the circle next to a user name to select an individual user. Click the circle  at the top of the list to select all users. To select multiple adjacent users, click the circle next to the first user and then press Shift + click the circle next to the last user.

   A user is selected when a check mark in the circle  accompanying the user name.
4. Beside **Applicable actions**, click **Activate/Deactivate**.
5. In the **Select a state** window, choose **Activate** or **Deactivate**.

   The user's status is changed.
6. Optional: In the Users list, click an individual user profile and click **Activate** or **Deactivate** to toggle the user's state.

## Delete a user (Local only)

Users in IBM Planning Analytics Workspace Local can be deleted.

### Before you begin

To administer users in IBM Planning Analytics Workspace Local, you must have the **Administrator** role. Your user profile shows your full name, user ID, role, and groups that you belong to.

### Procedure

1. Set the ENABLE_USER_IMPORT configuration parameter to FALSE.

**Note:** When you change the Planning Analytics Workspace configuration file, you must run `./scripts/paw` for your operating system or click **Start** in the Planning Analytics Workspace administration tool for your changes to take effect. Only services that are affected by the configuration change will restart.

For more information, see Planning Analytics Workspace configuration parameters.

2. On the **Welcome** page, click your user name, and then click **Administer**.

   **Note:** If you don't see the **Administer** option, then you are not logged in as administrator and you cannot delete users.

3. To delete users that you uploaded with the REMOVE directive, select the users with a status of **Not invited yet**.

   Click the circle next to a user name to select an individual user. Click the blue circle at the top of the list to select all users. To select multiple adjacent users, click the circle next to the first user and then press Shift + click on the circle next to the last user. A user is selected when a check mark appears in the circle accompanying the user name.

4. Beside **Applicable actions**, click **Delete users**.
5. In the **Confirm deletion** window, click **OK** to confirm the deletion or click **Cancel** to keep these users.

   A success message appears.

6. Click **OK**.

   The user is deleted.

## Delete multiple users (Local only)

Multiple users in IBM Planning Analytics Workspace Local can be deleted in bulk with a .csv file.

### Before you begin

To administer users in IBM Planning Analytics Workspace Local, you must have the **Administrator** role.

To delete multiple users from Planning Analytics Local, create a CSV file that contains one line for each user that you want to delete with the following information: Login ID, first name, last name, role, environment. Use the REMOVE directive.

**Note:** Text values in the CSV file, including column titles, roles, and environments, must be in English.

You can also export users to a CSV file and edit the file in a text editor to contain one line for each user that you want to delete. Change the ADD directive to REMOVE.

### Procedure

1. Set the ENABLE_USER_IMPORT configuration parameter to false.

   **Note:** When you change the paw configuration file, you must run `./scripts/paw` for your operating system or click **Start** in the Planning Analytics Workspace administration tool for your changes to take effect. Only services that are affected by the configuration change will restart.

   For more information, see Planning Analytics Workspace configuration parameters (https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_inst.2.0.0.doc/c_paw_config_file.html).

2. On the **Welcome** page, click your user name, and then click **Administer**.

   **Note:** If you don't see the **Administer** option, then you are not logged in as administrator and you cannot delete users.

3. Click the **Users** tab.
4. Click **Upload users**.
5. Drag the CSV file on to the **.csv** icon  , or browse to the file location and import the file.

   The file is uploaded and you see a success message.

6. Click **OK**.

## Understanding IBM Planning Analytics Local licensing

Planning Analytics on-premise licensing is based on one or both of two charge metrics.

- Processor Value Unit (PVU) for capacity-based licenses
- Authorized Users for user-based licenses

PVU totals are tracked using the IBM License Metric Tool (ILMT).

This section describes how to determine the number of authorized users in a Planning Analytics Local environment. An authorized user is a user that is granted access to one or more Planning Analytics databases. Each authorized user can access any number of Planning Analytics databases at the same time, but counts as only a single authorized user.

### *Determining authorized users on Planning Analytics Local*

To examine users and groups in a Planning Analytics database, the you must have a Modeler role in Planning Analytics Workspace and Administrator role on the Planning Analytics database.

You can track authorized users in Planning Analytics Workspace by examining the members in the `}Clients` control dimension for a specific Planning Analytics database. The `}Clients` dimension provides a list of users on the Planning Analytics database. The list in the `}Clients` dimension doesn't tell you if a user is in compliance with their license type or what license type the user is intended to have. You must track that detail manually.

1. Open a modeling workbench in Planning Analytics Workspace.
2. On the **Databases** tree, expand the database where you want to review authorized users.
3. Click **Control objects** > **Dimensions**.
4. Right-click the **}Clients** control dimension, then click **Edit dimension** to review the members of the dimension.

Authorized users of Planning Analytics Local must have one of the following license entitlements.

For Planning Analytics Advanced:

- IBM Planning Analytics Advanced
- IBM Performance Management Advanced

For Planning Analytics Local:

- IBM Planning Analytics Local Modeler (required)
- IBM Planning Analytics Local User
- IBM Planning Analytics Local Explorer
- IBM Performance Management User
- IBM Cognos Enterprise Planning TM1 Explorer

For Planning Analytics Express:

- IBM Planning Analytics Express Administrator (required)
- IBM Planning Analytics Express User
- IBM Cognos Express User
- IBM Planning Analytics Express TM1 User
- IBM Cognos Express Performance Management User

**Note:**

Users of Planning Analytics Advanced, Planning Analytics Local, and Planning Analytics Express cannot exist in the same environment. These three programs cannot share users, data, or servers. Each of the three must remain separate and distinct.

Access to cubes, administration and modeling capabilities, and Planning Analytics Workspace varies by license entitlement.

| | **Read cubes** | **Write cubes** | **Administration and Modeling** | **Planning Analytics Workspace** |
|---|---|---|---|---|
| **Planning Analytics Advanced** | | | | |
| IBM Planning Analytics Advanced | Yes | Yes | Yes | Yes |
| IBM Performance Management Advanced | Yes | Yes | Yes | Modeling Workbench and Planning Analytics Administration only. Add-on required for full access. |
| **Planning Analytics Local** | | | | |
| IBM Planning Analytics Local Modeler | Yes | Yes | Yes | Yes |
| IBM Planning Analytics Local User | Yes | Yes | No | Yes |
| IBM Planning Analytics Local Explorer | Yes | No | No | Yes |
| IBM Performance Management User | Yes | Yes | No | No. Add-on required. |
| IBM Cognos Enterprise Planning TM1 Explorer | Yes | No | No | No. Add-on required. |
| **Planning Analytics Express** | | | | |
| IBM Planning Analytics Express Administrator | Yes | Yes | Yes | Yes |
| IBM Planning Analytics Express User | Yes | Yes | No | Yes |
| IBM Cognos Express User | Yes | Yes | No | No. Add-on required. |
| IBM Planning Analytics Express TM1 User | Yes | Yes | No | Yes |
| IBM Cognos Express Performance Management User | Yes | Yes | No | No. Add-on required. |

- Do not assign users without administrator access to any of the administrator roles in the Planning Analytics database.

- Do not allow users without a license type that permits write access to belong to a group in the Planning Analytics database that provides write access to any cubes.

### *Determining read and write access to Planning Analytics cubes*

Read and write access to cubes in a Planning Analytics database is determined by the group membership of the authorized user.

An authorized user is a member of a group. Each group is granted READ or WRITE to objects, including cubes, in the Planning Analytics database.

Security access to cubes for each group is determined by examining the }CubeSecurity cube in the Planning Analytics database. The process script on this page examines the }CubeSecurity cube and }ClientGroups cube to produce a report that lists all cubes where a user has read access or greater. To use the script:

1. Create a new process on a modeling workbench.
2. Click the **Script** tab.
3. Copy and paste this script into the **Prolog** procedure.

```
DatasourceASCIIQuoteCharacter = '';
vReportFile = './model_upload/CubeAccessReport.txt';

vClientIndex = 1;
vClient = DIMNM( '}Clients', vClientIndex );
WHILE ( vClient @<> '' );
    IF ( vClient @<> 'Admin' );
        vClientAlias = ATTRS( '}Clients', vClient, '}TM1_DefaultDisplayValue' );
        IF ( vClientAlias @<> '' );
            ASCIIOutput( vReportFile, 'Authorized User: ' | vClientAlias );
        ELSE;
            ASCIIOutput( vReportFile, 'Authorized User: ' | vClient);
        ENDIF;
        vGroupIndex = 1;
        vGroup = DIMNM( '}Groups', vGroupIndex );
        WHILE ( vGroup @<> '' );
            vUsersGroup = CellGetS( '}ClientGroups', vClient, vGroup );
            IF ( vUsersGroup @= 'ADMIN' % vUsersGroup @= 'DataAdmin' % vUsersGroup @=
'SecurityAdmin' );
                ASCIIOutput( vReportFile, CHAR(9) | 'has ' | vUsersGroup | ' role');
            ENDIF;
            vGroupIndex = vGroupIndex + 1;
            vGroup = DIMNM( '}Groups', vGroupIndex );
        END;
        vCubeIndex = 1;
        vCube = DIMNM( '}Cubes', vCubeIndex );
        WHILE ( vCube @<> '' );
            vGroupMostAccess = '';
            vGroupIndex = 1;
            vGroup = DIMNM( '}Groups', vGroupIndex );
            WHILE ( vGroup @<> '' );
                vUsersGroup = CellGetS( '}ClientGroups', vClient, vGroup );
                IF ( vUsersGroup @<> '' % vUsersGroup @= 'ADMIN' % vUsersGroup @=
'DataAdmin' % vUsersGroup @= 'SecurityAdmin' );
                    vAccess = CellGetS( '}CubeSecurity',vCube, vUsersGroup );
                    IF ( vGroupMostAccess @= '' );
                        vGroupMostAccess = vAccess;
                    ELSEIF ( vGroupMostAccess @= 'READ' & vAccess @= 'WRITE' );
                        vGroupMostAccess = vAccess;
                    ELSEIF ( ( vGroupMostAccess @= 'READ' % vGroupMostAccess @= 'READ' ) &
vAccess @= 'ADMIN');
                        vGroupMostAccess = vAccess;
                    ENDIF;
                ENDIF;
                vGroupIndex = vGroupIndex + 1;
                vGroup = DIMNM( '}Groups', vGroupIndex );
            END;
            IF ( vGroupMostAccess @<> '');
                ASCIIOutput( vReportFile, CHAR(9) | vGroupMostAccess | ' access on cube ' |
vCube );
            ENDIF;
            vCubeIndex = vCubeIndex + 1;
            vCube = DIMNM( '}Cubes', vCubeIndex );
        END;
    ENDIF;
    vClientIndex = vClientIndex + 1;
```

```
    vClient = DIMNM( '}Clients', vClientIndex );
END;
```

No data source, variables, or parameters are required.

4. Save and run the process.

When the process runs successfully, a `CubeReportAccess.txt` file is written to the File Manager for the Planning Analytics database. To download and view the report:

1. Right-click the Planning Analytics database on the **Databases** tree of a modeling workbench, then click **File Manager**.
2. Select **CubeAccessReport.txt**, then click **Download**.

The report looks similar to this:

```
Authorized User: User10
    WRITE access on cube Compensation
    WRITE access on cube Employee
    READ access on cube FcstMethod
    READ access on cube Income Statement
    READ access on cube Income Statement Reporting
    WRITE access on cube Relative Time
Authorized User: User9
    WRITE access on cube Allocation Calculation
    WRITE access on cube Compensation
    WRITE access on cube Compensation Reporting
    READ access on cube ExternalFactors
    READ access on cube FcstMethod
    WRITE access on cube Income Statement
Authorized User: User8
    READ access on cube Asset Life
    READ access on cube Benefit Assumptions
    WRITE access on cube Capital
    WRITE access on cube Employee
    READ access on cube Exchange Rates
    WRITE access on cube GLTransactions
```

The report also indicates which users have any of the administrator roles on the Planning Analytics database (Admin, DataAdmin, and SecurityAdmin). Each Planning Analytics database also has a built-in Admin user that cannot be removed from the Admin role in the database. This user cannot be disabled and does not count as an authorized user when it is restricted from use. Users in the OperationsAdmin role who are not assigned to any other Admin roles in the database do not require a Modeler license entitlement. These are considered maintenance functions rather than administration or modeling functions.

### *Removing inactive users from a Planning Analytics Local database*

Users are not automatically removed from a Planning Analytics database - even if the database is configured for Cognos or LDAP authentication.

You must manually remove users that are no longer licensed to use Planning Analytics from all Planning Analytics database. For details on removing users, see https://www.ibm.com/docs/en/planning-analytics/2.0.0?topic=workspace-object-security.

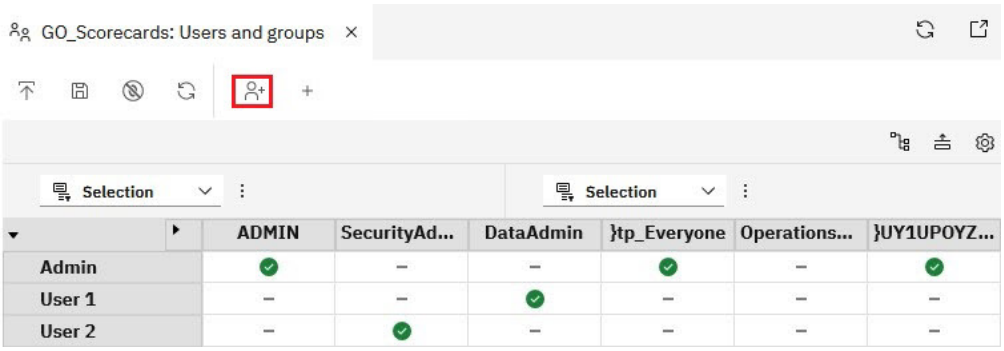
## Create users in a Planning Analytics Local database

You can create users in a Planning Analytics Local database using the **Users and groups** editor.

The user is added to the Planning Analytics Local database (TM1 server) only, not to IBM Planning Analytics Workspace. If you want to use the user for authentication, the user must also exist in the IBM Planning Analytics Workspace.

### Procedure

1. In a modeling workbench, right-click a database in the **Databases** tree, and then click **Manage users and groups**.
2. In the **Users and groups** editor, click the **Create new user** icon , as shown in the following image:




3. In the **Create new user** dialog box, type the user name and password, and click **Add**.

   The new user name appears at the end of the list of existing users.

   **Note:** To manage the users in the database, right-click the user name and click any of the menu options, such as **Remove user(s)** or **Change password**.

### Results

The user is added to the Planning Analytics Local database.

## Administer groups

Groups can be used to help manage and organize large numbers of users. Administrators can create, modify, or delete groups by using CSV files.

### About this task

To create multiple groups, delete existing groups, or to add users to existing groups, create a CSV file and click **Upload groups**. To create one group at a time, click **Create** and enter the required information. You can also click **Export groups** to save and edit a correctly formatted CSV file that you can use to make changes to existing groups.

### Administer groups in Planning Analytics Workspace

### Procedure

1. To create one group at a time:
   a) On the Home page, click the **Administration** tile.

      **Note:** You can't create a group with the name `Everyone` because Planning Analytics Workspace uses that as a default group when there are no groups defined.

   b) Click **Users & Groups**.
   c) Click **Groups**.
   d) Click **Create a new group** +.
   e) Enter a **Name** and **Description** for the group, then click **Submit**.

      The new group opens on the **Users & Groups** page.

   f) Click **Manage**.
   g) Select the users you want to add to the group, then click **Save**.
2. To create multiple groups, delete existing groups, or to add users to existing groups, create a CSV file that contains the groups to create or update.

   The format of the CSV file is described in the user interface. Use a header row as the first row of your CSV file. The CSV file must contain one line for each action with the following information: Group, Login ID, and Directive (ADD or REMOVE). The Login ID must be a valid user in this environment.

The following example .csv file adds four groups (`Test`, `Development`, `Quality`, `Management`) and adds the Login ID `admin` to each group.

```
Group,Login Id,Directive
Test,admin,ADD
Development,admin,ADD
Quality,admin,ADD
Management,admin,ADD
```

**Tip:** You can leave the Login ID blank (with no spaces) to create an empty group with no users in it.

a) On the Home page, click the **Administration** tile.

b) Click **Users & Groups**.

c) Click **Groups**.

d) Click **Upload groups from a csv file**.

e) Drag and drop your .csv file onto the **Drop your .csv file here** region or click the region and browse for your file.

# Export users

If you are an administrator, you can export all users in your environment to a CSV file. The exported CSV file can be updated and imported back into IBM Planning Analytics Workspace, which makes it easy to update roles and environments of users with a single bulk upload.

## About this task

You can export users in a .csv file and save the file to your Downloads folder as `all-users.csv` or open it in the editor and save it with a file name of your choice.

The .csv file is formatted so that you can edit it and upload users back into your environment with any changes that you want to apply. Columns in the exported .csv file match the format that is required for importing users. The values in the `Directive` column are hardcoded as ADD, which can be used to update existing users. The `Last Login` column shows the date and time of the most recent login for each user, but this column is ignored when importing users.

Each row in the .csv file represents a user in an environment. If a user is in multiple environments, there will be multiple rows for that user.

| | A | B | C | D | E | F | G |
|---|---|---|---|---|---|---|---|
| 1 | Login Id | First Name | Last Name | Role | Environment | Directive | Last Login |
| 2 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-06-16T13:15:41Z |
| 3 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-06-22T22:36:26Z |
| 4 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-06-23T13:44:17Z |
| 5 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-06-23T15:33:16Z |
| 6 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-06-23T18:35:09Z |
| 7 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-06-23T19:06:10Z |
| 8 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-06-24T13:23:02Z |
| 9 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-09-21T16:02:35Z |
| 10 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2022-09-22T12:33:14Z |
| 11 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-01-03T19:36:17Z |
| 12 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-02-06T18:36:23Z |
| 13 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-02-06T18:36:29Z |
| 14 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-02-09T16:42:18Z |
| 15 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-02-17T20:25:18Z |
| 16 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-04-21T12:15:59Z |
| 17 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-04-25T18:27:53Z |
| 18 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-04-26T15:42:16Z |
| 19 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-04-26T15:42:16Z |
| 20 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-04-26T15:45:05Z |
| 21 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-04-26T15:45:05Z |
| 22 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-04-27T17:32:59Z |
| 23 | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | [redacted] | 2023-04-27T17:33:33Z |

### Export users in Planning Analytics Workspace

#### Procedure

1. On the Planning Analytics Workspace home page, click the **Administration** tile.
2. Click **Users & Groups**.
3. Click the **Users** tab.

   **Note:** Selecting users has no impact. Clicking **Download users** always exports all users in the current environment.

4. Click **Download users**  .

## Change a user's role

You can change the role for one or more users. In the user interface, you can select a single user and change their role. Or, you can export users to a CSV file, make changes to roles and then upload those changes.

**Note:** In a user's profile, you can see only the role that a user has in the current environment. If they belong to another environment with a different role, you must log in that environment to see that role reflected in the user profile.

When you make changes to the roles of users in the current environment using a CSV upload, you will see the changes reflected in the users' profiles in the current environment.

If you make changes to the roles of users in other environments using a CSV upload, you must log in to those environments to see the changes reflected in the users' profiles.

### Before you begin

When you are an account administrator for multiple organizations, you must use https://myibm.ibm.com to manage users in any organization other than the first organization to which you were added.

### About this task

The following roles are available:

**Consumer**

Consumers can open books and views and other content that is shared with them.

Consumers cannot create their own books and views, but they can share content that is shared with them, with consumer rights only.

Consumers can delete books and views if they have **Full control** permission set for those books and views.

**Note:** When you log in to Planning Analytics Workspace from an iPad, you are always a Consumer, regardless of your actual role. For more information, see “Accessing Planning Analytics Workspace from Apple iPad” on page 427.

**Analyst**

Analysts have the rights of a consumer, plus the following abilities:

Analysts can create, edit, and share books and views.

Analysts can delete books and views if they have **Full control** permission set for those books and views.

**Modeler**

Modelers have the rights of an analyst plus the following abilities:

They can create and share content, and edit content that is shared with them.

Modelers can create, edit, and delete dimensions and hierarchies. They can add, delete, cut, paste, copy, move, and sort members and their attributes in a hierarchy.

For more information, see “The modeler role” on page 718.

**Administrator**

Administrators have all the rights of a modeler, plus the following abilities:

Administrators can see all content in the workspace.

Administrators can set permissions for a book.

**Note:** At least one user in your organization must have the administrator role. If your organization has only one administrator, this user cannot be deleted or assigned to another role. Administrators can assign roles to users and only an administrator can change another user's role to administrator.

By default, the administrator role is assigned to the first user in your organization's Planning Analytics Workspace account. In Planning Analytics Workspace on cloud, this first administrator is also known as the *subscription administrator*. Only the subscription administrator can add new users.

### Change a user's role in Planning Analytics Workspace

#### Procedure

1. On the Home page, click the **Administration** tile.
2. Click **Users & Groups**.
3. Click the **Users** tab.
4. To change the role for a single user:
   a) Select the user on the **Users** tab.
   b) Select a new role on the **Details** tab.
   c) Click **Apply**.
5. To change multiple users to the same role:
   a) Select the users on the **Users** tab.
   b) On the selected items pane, click **Set role**.
   c) Select a role for the users and click **Apply**.

## Change a user's environment (Cloud only)

A user's environment may need to be changed as their work needs evolve or change over time. An administrator can use a CSV file or the user interface to change a user's environment.

### Before you begin

When you are an account administrator for multiple organizations, you must use https://myibm.ibm.com to manage users in any organization other than the first organization to which you were added.

### About this task

Users in IBM Planning Analytics Workspace can have access to one or more environments. For example, a user can access both your non-production and production environments. In the user interface, you can select a single user and change their environment. Or, you can export users to a CSV file, make changes to environments and then upload those changes.

You can change the environment for one or more users at time.

### Results

If a user has multiple environments assigned to it, they will be prompted to select one the next time they log in.

### Change a user's environment in Planning Analytics Workspace

#### Procedure

1. On the Home page, click the **Administrator** tile.
2. Click **Users & Groups**.
3. Click the **Users** tab.
4. To change environments for a single user:
   a) Select the user on the **Users** tab.
   b) On the adjacent user detail pane, click the **Environments** tab.
   c) Click **Manage**.
   d) Select the environments for the user, then click **Save**.
5. To change environments for multiple users:
   a) Select the users on the **Users** tab.
   b) On the selected items pane, click **Set environment**.
   c) Select the environments for the users and click **Save**.

## Copy and deploy assets with Lifecycle Management in Planning Analytics Workspace

Assets such as books, views, folders, applications, and plans can be copied and deployed by an administrator from a source environment to target environment using Lifecycle Management.

If you are an administrator, you can deploy any assets that you have permission to view from a source environment to a target environment by clicking **Lifecycle Management** on the **Administration** page in IBM Planning Analytics Workspace. An asset can be a book, view, folder, modeling workbench, application, or plan. Websheets are not supported as an asset type.

Customers on cloud are provisioned with multiple products in multiple environment. They can have a development environment and a production environment. As an administrator, you can build content in a development environment and then deploy it to your production environment when you are ready. If the target environment is available, you can deploy assets in one step. If the target environment is a IBM Planning Analytics Local instance, you can export a snapshot of assets from your source environment and import the snapshot in the target environment.

You must have at least **View** permission for an asset to see it in Lifecycle Management. You are not shown assets that you don't have permission to view even if they exist on your source environment.

You can also create a snapshots of assets as a backup without deploying it, so that you can restore an earlier version of your assets whenever you want.

**Remember:** A snapshot is a collection of assets at a single point in time. Lifecycle Management validates that the content of the snapshot that you are creating can be deployed to the target environment. When you deploy a collection of assets from the source environment to the target environment, the contents of the assets are deployed as well. If a book contains a websheet, the websheet must be deployed manually.

### Prerequisites

To deploy assets with Lifecycle Management, you must be logged in to IBM Planning Analytics Workspace as an administrator.

When you are an administrator, the **Administration** tile is available on the Planning Analytics Workspace home page. Click the tile to open the Administration page, where you can then click the **Lifecycle Management** tile to create and manage snapshots and deploy assets.

For more information about the administrator role, see “User roles” on page 415.

## Limitations

The following limitations apply to Lifecycle Management in IBM Planning Analytics Workspace.

- You must have at least **View** permission for an asset to see it in Lifecycle Management. You are not shown assets that you don't have permission to view even if they exist in your source environment.
- You can deploy assets that exist in the Shared folder or in your Personal folder. You cannot migrate assets that exist in other users' Personal folders.
- Importing a snapshot from a newer version of Planning Analytics Workspace into an older version of Planning Analytics Workspace is not supported. For example, you should not import a snapshot that was created in Planning Analytics Workspace version 2.0.90 into version 2.0.89. Though Lifecycle Management does not prevent such an import, you might experience unexpected results due to the difference in book specifications between versions.
- Websheets are not supported as an asset type. This includes websheets within a plan or application. Websheets must be manually moved to the target environment and placed in the appropriate folder structure.
- If you are deploying a book that contains multiple missing databases, when you remove the book from the snapshot all of the relevant warnings for that book are not removed.
- If you create a circular mapping of databases, all databases are mapped to a single database. For example, mapping 24Retail to 2003cert and 2003cert to 24Retail maps all databases to 24Retail.
- If a step in a plan includes a contributor or approver group that is not available on the target environment, the group is removed from the step when the snapshot is deployed. You must manually update the step to use a valid group on the target environment.

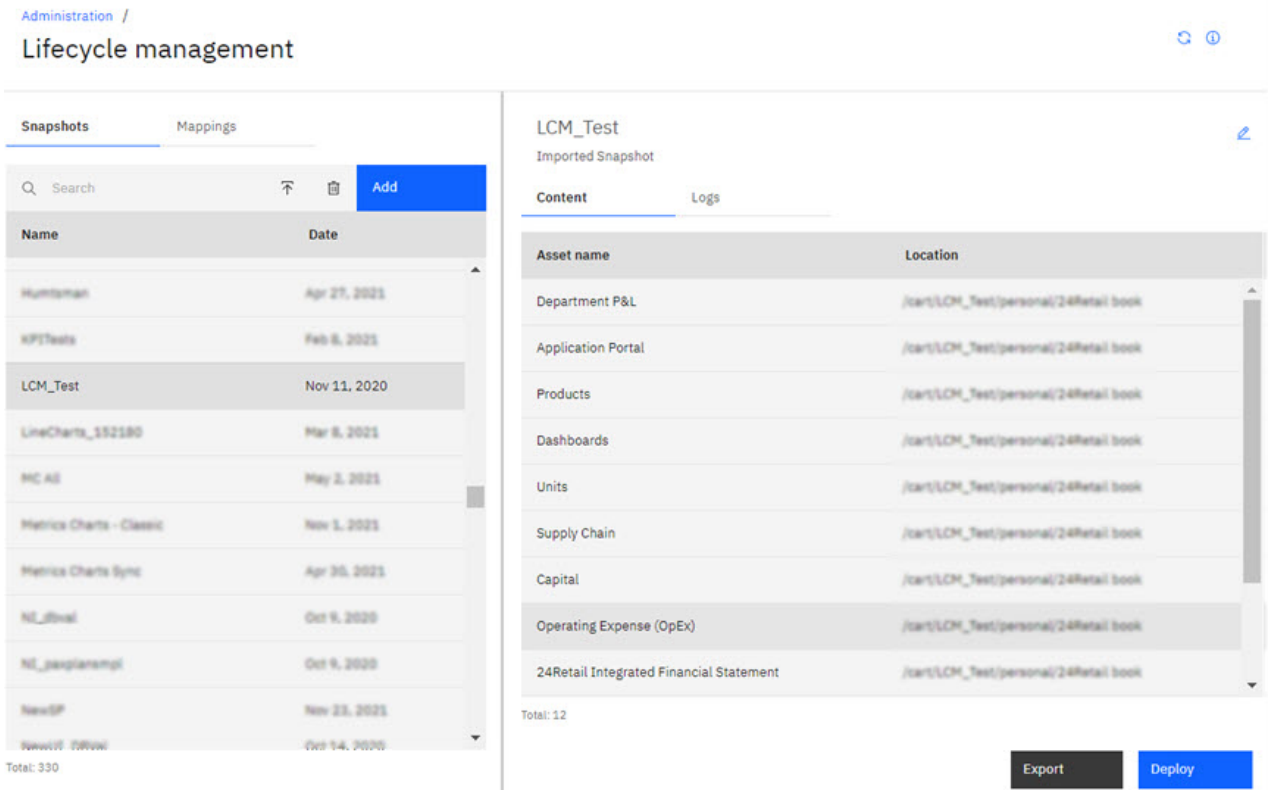
## Get started

On the IBM Planning Analytics Workspace **home** page, click the **Administration** tile to open the **Administration** page. To copy and deploy assets, click **Lifecycle Management** on the **Administration** page.

The **Lifecycle Management** page has two tabs. The **Snapshots** tab lists all the snapshots available on your database. The **Mappings** tab provides access to the current database mappings from your source to your target environment and lets you create new mappings.

**Note:** The Mappings tab is not available on Planning Analytics Workspace Local, it is available only on Planning Analytics Workspace on Cloud.

When you click a snapshot name, you can view the assets that are included in the snapshot on the **Content** tab or review the logs for the snapshot on the **Logs** tab.




# Create a snapshot

You can add multiple assets to a snapshot so that you can deploy the assets to a target environment. You can also optionally export and save the snapshot as a .gz archive. Snapshot archives can be imported into Lifecycle Management.

## Procedure

1. Click **Add** on the **Snapshots** tab of the Lifecycle Management page.
2. Enter a **Name** for the snapshot.
3. Optionally, enter a **Description** of the snapshot.
4. Click **Next**.
5. Select the assets from the **Available** list that you want to include in the snapshot. You can include assets from either or both the Shared and Personal folder, but you can only select assets from one of those folders at a time.
   a) Click **Shared** or **Personal** to select assets from either folder.

      When you click either **Shared** or **Personal**, the **Search** and **Filter** options become available to help you locate assets. You can type an asset name in the search box or click **Filter** to narrow the display of assets to a selected asset type (books, views, or folders).

   b) Select the assets you want to include in the snapshot.

      When you select a view or book from the Available list, it is added to the Selected list. When you select a folder from the Available list, all assets within the folder (and any sub-folders) are added to the Available list as individual assets. When you select a plan or application from the Available list, all assets within the plan or application are added to the Available list as required assets of the plan or application.

   c) If you want to remove an asset from the Selected list, click the **Remove from selection** icon next to the asset.
6. Click **Create**.

### Results

The snapshot is created and selected in the snapshots list. The **Details** panel shows the name and location for all assets in the snapshot.

In both the snapshots list and the **Details** panel, you can see the version of IBM Planning Analytics Workspace that the snapshot was created in.

**Note:** This feature is only available in IBM Planning Analytics Workspace 2.0.96 or later. A snapshot can display versions 2.0.90 or later. If the snapshot version is not displaying, you may need to export the snapshot and import it again.

The snapshot is now available to deploy.

## Configure database mappings

You can pre-configure the mapping of a database in your source environment to a database in your target environment so that you can deploy your assets faster.

### About this task

Pre-configured database mappings streamline common scenarios, such as development to production environment mappings. The mappings are stored for each target environment not for each user. You must have permission to view assets in a database in the source and target environments to set up the database mappings.

**Note:** You can map multiple databases in the source environment to one database in the target environment. However, you can't map a database in the source environment to multiple databases in the target environment.

- If you deploy assets from a source database to a target database that doesn't exist and you have set up a database mapping, the mapping is used to set the database in the target environment so that you don't need to select it manually.
- If you deploy assets from a source database to a target database that exists and this contradicts the database mappings, the mapping is ignored and the assets are migrated as specified.
- If you migrate assets from a source database to a target database that doesn't exist and doesn't have a mapping set up, the database must be selected manually.

### Procedure

1. Click the **Mappings** tab of the Lifecycle management home page.
2. Click the **Target tenant** for which you want to create mappings.
3. Click **Add** on the mappings pane.
4. Click a **Select a database** under **Database in <source tenant>**. Select the database on the source that you want to map to the target tenant.
5. Click a **Select a database** under **Database in <target tenant>**. Select the database on the tenant to which you want the source database to map.
6. Repeat steps 3 through 5 for all mappings you want to establish.

# Deploy a snapshot

After you create a snapshot, you can deploy it to a selected environment.

### About this task

See “Create a snapshot” on page 907 for information on creating a snapshot of assets.

### Procedure

1. Click the snapshot you want to deploy on the **Snapshots** tab of the Lifecycle Management home page.
2. Click **Deploy**.
3. Review the assets that will be deployed in the snapshot, then click **Next**.
4. Select the folder (**Personal** or **Shared**) on the target environment where the snapshot will be deployed.

   If your snapshot includes plans or applications, you must deploy to the Shared folder.
5. Enter the **Folder** name where the assets will be deployed on the target environment.

   The folder name you specify is created within either the Personal or Shared folder on the target environment, according to your selection in step 4.
6. For Planning Analytics Workspace on Cloud, select the **Target tenant** for the deployment. The snapshot is deployed on the tenant you select.

   For Planning Analytics Workspace Local, there is no option to select a target tenant, as there is no tenancy in Planning Analytics Workspace Local.
7. Click **Next**.

   The **Database mapping** page shows the database mappings for databases used by assets in your source environment to databases on the target environment.

   For Planning Analytics Workspace Local, the Database mapping page shows the databases associated with your assets mapped to the same database name.

   For Planning Analytics Workspace on Cloud, the Database mapping page shows the pre-defined database mappings between your source and target tenants. If a database mapping is not defined for a database on your source environment, the same database name is used on the target tenant.

   You can accept the database mappings as they are presented or you can modify them.

   **Note:** You might see a warning icon next to a target database. This indicates that the database is not currently available on the target tenant. It's possible that the database is not running or has not yet been created. If you intend to start or create the database after deployment, you can proceed; the deployment will continue.

   If you do not intend to start or create the database after deployment, consider changing the mapping from the source tenant database to a database that is available on the target tenant.
8. To change a mapping between a database on the source tenant and a database on the target tenant, click ⌄ next to the target database and select a new database.
9. Click **Deploy**.

## Export a snapshot

You can export a snapshot to your local file system. The snapshot can be saved as a backup or imported into another Planning Analytics Workspace target environment.

### Procedure

1. Click the snapshot you want to export on the **Snapshots** tab of the Lifecycle Management home page.
2. Click **Export**.

### Results

A `.gz` archive is saved in your `Downloads` directory, using the snapshot name.

## Import a snapshot

You can import a snapshot from your local file system into Lifecycle Management.

### Procedure

1. Click the **Import snapshot** icon on the **Snapshots** tab of the Lifecycle Management home page.
2. Drag the *snapshot*.gz file onto the drop zone or click the drop zone to browse for the snapshot.
3. If a snapshot exists in your source environment with the same name as the file that you are importing, you're asked to choose one of the following options:
   a) Import and keep both snapshots by renaming the snapshot that is being imported. The existing snapshot remains unchanged and both snapshots appear in the snapshot list.
   b) Import and replace the existing snapshot. The existing snapshot is deleted.
   c) Cancel your import. The existing snapshot remains unchanged.
4. Click **Import** to import a snapshot. Select a snapshot from your local file system. Note: You must choose a file with the GZ file type (.gz). If a snapshot exists in your source environment with the same name as the file that you are importing, choose one of the following options: Import and keep both snapshots by renaming the snapshot that is being imported. The existing snapshot remains unchanged and both snapshots appear in the snapshot list. Import and replace the existing snapshot. The existing snapshot is deleted. The imported snapshot appears in the snapshot list. Cancel your import. The existing snapshot remains unchanged.

## Rename or delete a snapshot

You can change the name and description of a snapshot. You can also delete a snapshot.

### Procedure

1. Click the snapshot you want to rename or delete on the **Snapshots** tab of the Lifecycle Management home page.
2. To rename the snapshot, click **Rename snapshot** on the snapshot details pane.
   a) Enter a new name for the snapshot, and optionally enter a revised description.
   b) Click **Save**.
3. To delete a snapshot, click **Delete snapshot** on the **Snapshots** tab. Click **Delete** when prompted for confirmation.

## View a snapshot log

You can view a log of actions for a selected snapshot.

### Procedure

1. Click the snapshot on the **Snapshots** tab of the Lifecycle Management home page.
2. Click the **Logs** tab on the snapshot details pane.

   You'll see a list of all actions performed on the snapshot, with a corresponding date and user for each action.

   You can click any log heading to sort the log.

# Manage features

When some new features are introduced to Planning Analytics Workspace, you can choose to enable or disable the features in your environment. You can manage when and how your users are exposed to new functionality.

## About this task

New features that are subject to management by administrators are disabled by default. You have the option of enabling some features when they are introduced in Planning Analytics Workspace. However, after the feature has been available for several release cycles, it becomes permanently enabled and is no longer subject to management by an administrator. The release in which the feature becomes permanent is referred to as the **Enablement Release** and it is clearly indicated for each feature.

## Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace home page.
2. Click the **Features and Configuration** tile.

   **Note:** In Planning Analytics as a Service, this tile is titled **Features**.
3. Click the **Features** tab.

   The **Features** tab lists all the features that you can manage and identifies the current release of Planning Analytics Workspace that you are using.

   Each feature is identified by its name. Click the name to view a brief description and the **Enablement Release** for the feature.
4. To enable or disable a feature, click the **Status** toggle and confirm your selection when prompted.

# Disable select data spreading methods on Planning Analytics Workspace Local

You can configure your Planning Analytics Workspace Local environment to disable any combination of the **Repeat**, **Repeat leaves**, **Equal leaves**, and **Straight line** data spreading methods.

## About this task

When you disable a data spreading method, the method cannot be applied in a cube view or in a single cell widget. The method is grayed-out and cannot be selected in the **Data spread options** window and the spreading method cannot be applied using data spread keyboard shortcuts.

**Important:** This procedure disables data spreading only in a cube view or in a single cell widget in Planning Analytics Workspace. If Planning Analytics for Microsoft Excel is also available in your local environment, these settings have no impact on data spreading in Planning Analytics for Microsoft Excel.

## Procedure

1. Open the Planning Analytics Workspace Local configuration file.

   The configuration file is located in the `<planning_analytics_install_dir>\config` directory and is named `paw.ps1` (Windows) or `paw.env` (Linux).
2. Add the **`DISABLED_SPREAD_OPTIONS`** parameter to the configuration file and specify the data spreading methods you want to disable.

   The methods you can specify for this parameter are:

   - Repeat
   - RepeatLeaves
   - EqualLeaves

- StraightLine

  If you specify multiple data spreading methods, separate each method with a comma and enclose the entire parameter value string in quotation marks as shown in the example.

3. Save the configuration file.
4. Run `./scripts/paw.ps1` (Windows) or `./scripts/paw.env` to apply the configuration changes.

**Example**

To disable the Repeat, Repeat leaves, Equal leaves, and Straight line data spreading methods on Windows, add this line to the `paw.ps1` configuration file.

```
$env:DISABLED_SPREAD_OPTIONS="Repeat,RepeatLeaves,EqualLeaves,StraightLine"
```

To disable the Repeat, Repeat leaves, Equal leaves, and Straight line data spreading methods on Linux, add this line to the `paw.env` configuration file.

```
export DISABLED_SPREAD_OPTIONS="Repeat,RepeatLeaves,EqualLeaves,StraightLine"
```

# Configure the Planning Analytics Workspace environment

Use the **Configuration** tab on the **Features and Configuration** page to configure various aspects of the Planning Analytics Workspace on Cloud environment. This option is not available on IBM Planning Analytics as a Service.

## Configure an inactivity timeout for Planning Analytics on Cloud

Planning Analytics Workspace on Cloud enforces a configurable inactivity timeout. When a user is inactive for the period of time specified by the Planning Analytics Workspace administrator, the user is disconnected from the Planning Analytics database.

### About this task

This inactivity timeout is mandatory and is applicable to all Planning Analytics Workspace on Cloud users, as well as Planning Analytics for Excel users.

In addition to this configurable inactivity timeout, Planning Analytics Workspace also enforces a session timeout. For details, see Connection timeouts in Planning Analytics Workspace on Cloud.

For details on configuring timeouts in Planning Analytics Workspace Local, see Configuring parameters for Planning Analytics Workspace Local.

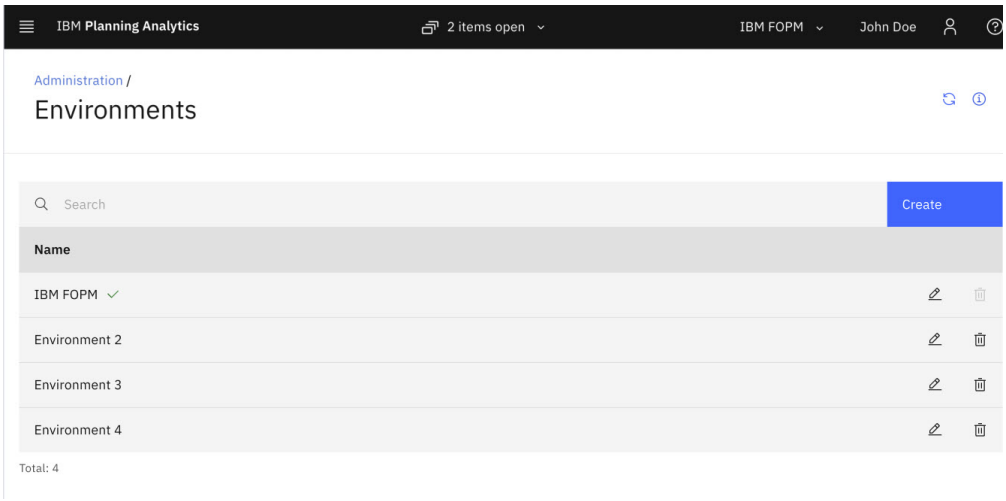
To set an inactivity timeout:

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace home page.
2. Click the **Features and Configuration** tile.
3. Click the **Configuration** tab.
4. Click the **Inactivity timeout** configuration parameter.
5. Use the **Hours** and **Minutes** controls to set an inactivity timeout period. The minimum timeout period is 30 minutes and the maximum period is 24 hours.

# Create and edit multiple Planning Analytics Local environments (Local only)

You can manage multiple environments in Planning Analytics Administration Local.




To create a new environment:

1. Enable multiple environments by adding the following to the `paw.env` on Linux or `paw.ps1` file on Windows:

   ```
   export ENABLE_MULTI_ENV="true"
   ```

2. Restart Planning Analytics Workspace.
3. Click the **Administration** tile on the Planning Analytics Workspace home page.
4. Click the **Environments** tile.
5. Click **Create**.
6. Enter an **Environment name**, **Spreadsheet service URL**, and **Admin host**, then click **Create**.

   The **Admin host** that you specify determines which databases are available in the environment. Only the databases registered on the specified admin host are available in the environment.

   You can choose to leave the **Spreadsheet service URL** and **Admin host** values blank when you create an environment. Entries for these values are then automatically used from the `paw.env` file on Linux or `paw.ps1` file on Windows.

To edit or delete an existing environment, click the environment name on the **Environments** page, then click the appropriate icon.

You can assign an individual user to any combination of available environments. Users can have different roles in different environments. For example, a modeler in Environment 1 could be assigned the analyst role in Environment 2.

When you log in to Planning Analytics Local and have access to multiple environments, an environment switcher is available on the Planning Analytics Workspace Local home page. adjacent to your user name. You can click the switcher to switch between environments that are available to you.



**Tip:** If you have multiple environments available to your users, consider applying a custom theme to each environment to make it easier to visually differentiate between environments.

# Monitor and administer databases

IBM Planning Analytics Administration includes the ability to monitor and administer your databases from the **Administration** page. You must be an administrator to see the Administration tile on the Planning Analytics Workspace home page and to use the **Administration** page.

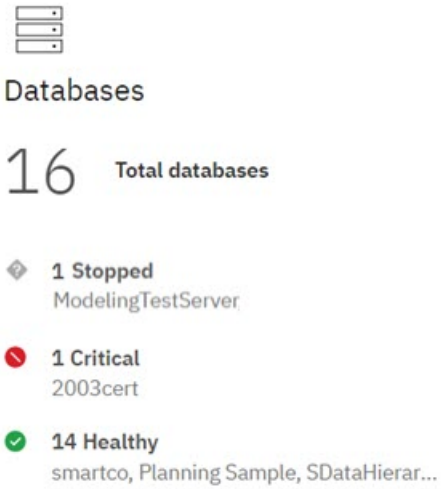
**Note:** To use Planning Analytics Administration on Planning Analytics Workspace Local, you must install and configure the Planning Analytics Administration agent wherever you have installed the Planning Analytics database. For more information, see "Install and configure the Planning Analytics Administration agent (local only)" on page 919.

There are some variations in the capabilities and procedures when monitoring and administering databases on Planning Analytics Administration. For more information, see "Manage databases on Planning Analytics as a Service" on page 957.

To open the **Administration** page, click the **Administration** tile on the **home** page.

## Monitor and administer databases in Planning Analytics Workspace

The Databases tile on the **Administration** page shows the total number of databases in your environment, along with a status summary for all databases.




Databases are identified as being in one of these four states:

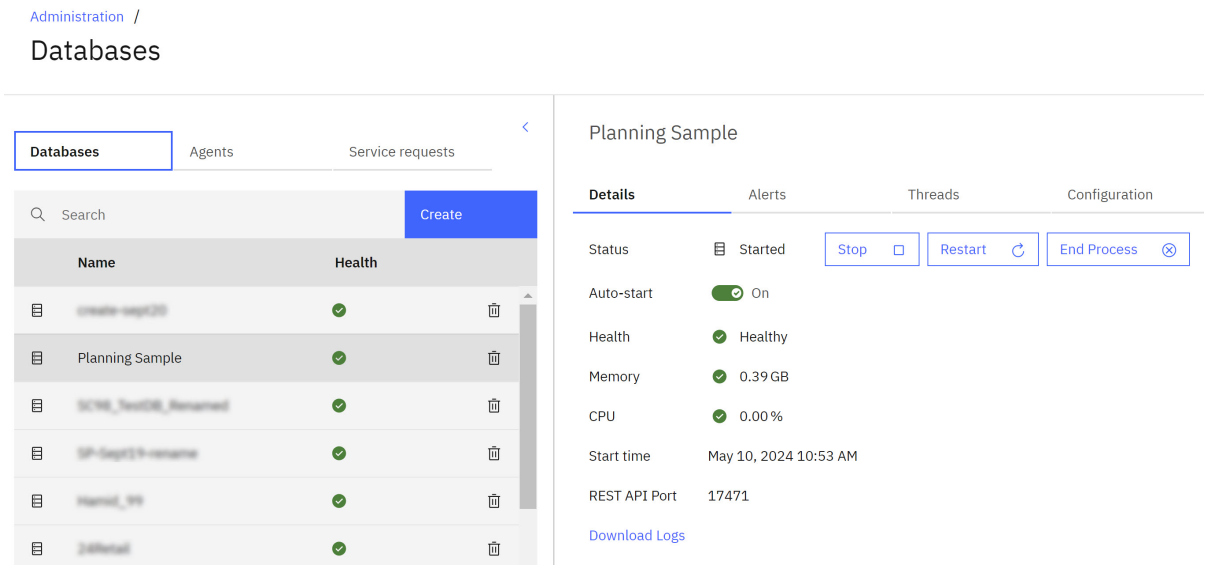
- Stopped - The database is nor running.
- Critical - When any single database resource reaches the critical threshold, the entire database is in a critical state.
- Concern - When any single database resource reaches the warning threshold, and there are no resources have reached the critical threshold, the entire database is in a warning state.
- Healthy - When all database resources are below the critical and warning thresholds, the entire database is in a healthy state.

Click the **Databases** tile to open the **Databases** page. This page shows a list of all the databases in your environment. For each database, you can see if it is running or stopped, the database name, and the database health.

Stopped databases are identified by the icon, while running databases are identified by the icon.

You can sort databases by running/stopped status, name, or health.

You can click any database name on the Databases list to view details for the database. The **Start time** or **Stop time** reported for the database reflects the most recent database start or stop initiated directly through the Planning Analytics Workspace Administration page. If a database is started or stopped in any other way, such as through IBM Cognos Configuration or an API call, those times are not reported.




In Planning Analytics Workspace Local, the Details page also indicates the agent on which the database is running.

## Connection timeouts in Planning Analytics Workspace on Cloud

Planning Analytics Workspace on Cloud enforces two connection timeouts.

Planning Analytics Workspace on Cloud enforces an inactivity timeout. When a user is inactive for a period of time specified by the Planning Analytics Workspace administrator, the user is disconnected from the Planning Analytics database. This inactivity timeout is mandatory and applicable to all Planning Analytics Workspace on Cloud and Planning Analytics for Excel users.

When an inactivity timeout does occur, what is the impact on the user?

- Workspace end user state will be lost.
- Authoring of Books and Plans/Applications that are not saved will be lost.

Note that the inactivity timeout is applicable only to Planning Analytics Workspace on Cloud. For details on configuring timeouts in Planning Analytics Workspace Local, see Configuring parameters for Planning Analytics Workspace Local.

The inactivity timeout also applies to Planning Analytics for Excel users. If a Planning Analytics for Excel user is inactive for a period of time specified by the Planning Analytics Workspace administrator, they will be disconnected and must log back in. However, there is no loss of state when a user is disconnected from Planning Analytics for Excel due to inactivity.

In addition to the inactivity timeout, Planning Analytics Workspace on Cloud enforces a 24 hour session timeout. This session timeout disconnects a user 24 hours after their most recent login and requires the user to log back in to Planning Analytics Workspace. The session timeout is enforced regardless of user activity. If a user is active continuously and never encounters an inactivity timeout, the session timeout is still triggered 24 hours after the most recent login.

## What are the differences between Planning Analytics Administration on premises and on cloud?

The following database administration capabilities **are** available in Planning Analytics Workspace Local:

- “Monitor and manage database threads” on page 927
- “Start and stop databases” on page 940
- Download database log files
- Download the latest version of the Planning Analytics Administration agent

The following capabilities **are not** available in Planning Analytics Workspace Local:

- Satellite Connector
- Planning Analytics for Microsoft Excel download
- More than one Planning Analytics Administration agent
- Database creation
- System resources, thresholds, and server disk usage

# Configure system monitoring

You can configure the interval at which database status is refreshed on the **Databases** page, as well as configure system resource thresholds and alerts.

## Set agent resource thresholds and alerts

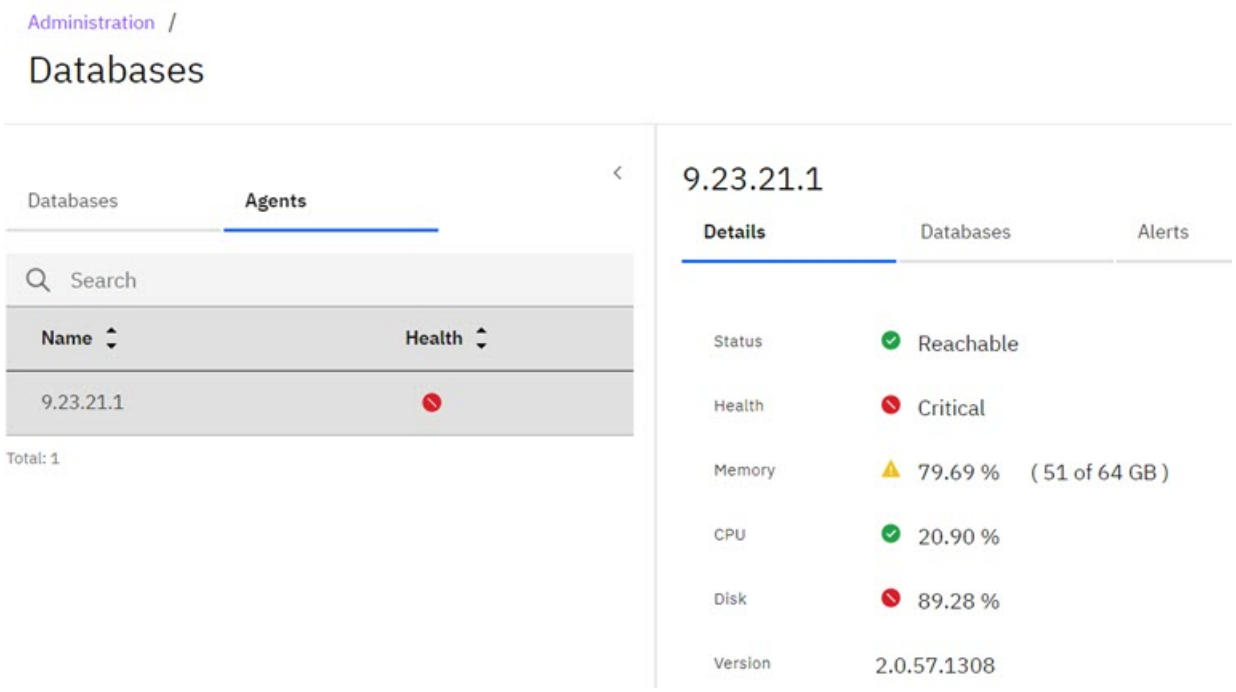
You can configure threshold settings that define agent resource status by multiple metrics. You can also configure alerts to send notification to yourself and other users when any of the system resources reaches a critical state.

### About this task

When you set threshold limits, you define the parameters for the safe, concern, and critical states of these system resource metrics:

- Memory
- CPU
- Disk

In Planning Analytics Workspace, these states are displayed on the Databases page. A check mark in a green circle indicates a resource in a healthy state. A yellow triangle with an exclamation mark indicates a resource in a concern state. A slash in a red circle indicates a resource in a critical state.



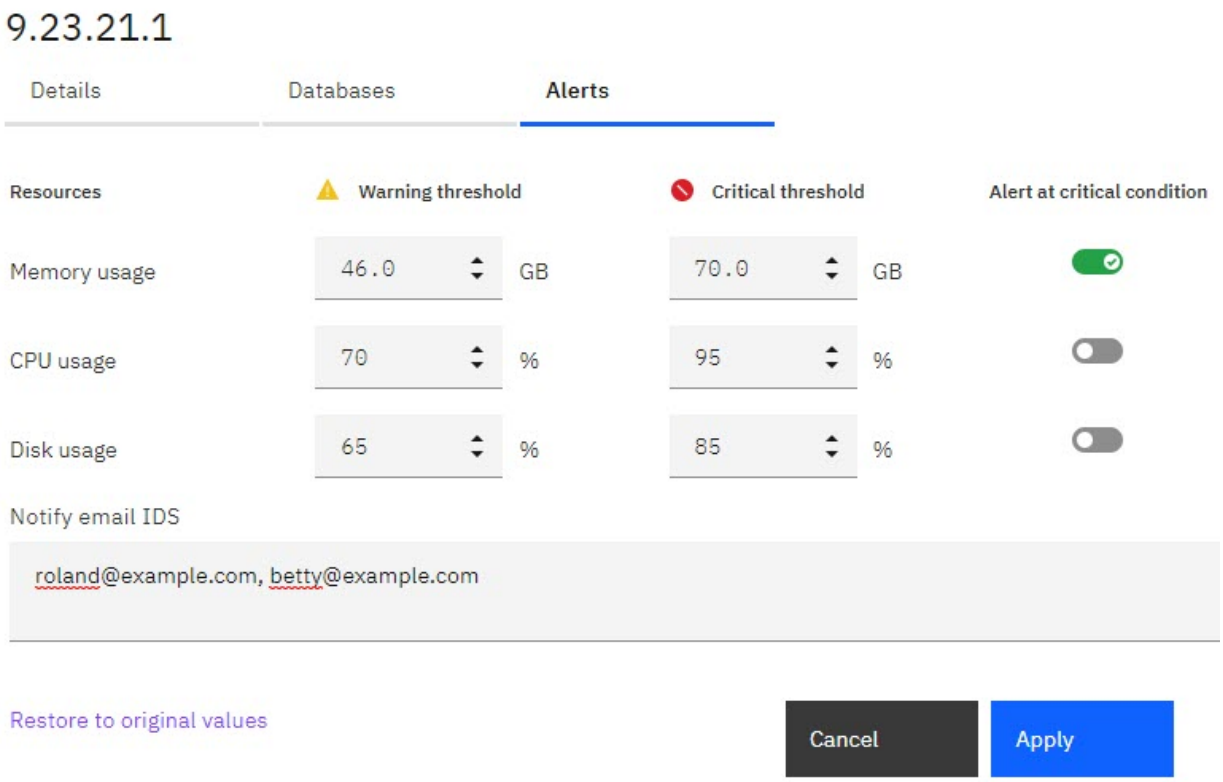

For Planning Analytics Administration on Cloud, threshold and alert settings apply to your entire environment.

For Planning Analytics Administration Local, you must configure threshold and alert settings for each agent in your environment. Unique system resource metrics are displayed for each agent in your environment.

### *Set agent resource thresholds and alerts in Planning Analytics Workspace*

## Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace Home page.
2. Click **Databases**.
3. Click the **Agent** tab on the **Databases** page.
4. Click the specific agent for which you want to set thresholds and alerts.
5. Click the **Alerts** tab.




6. For each system resource, set the **Warning threshold** and **Critical threshold**.

   When resource usage falls below the Warning threshold, the resource is in a healthy state. When resource usage is at or above the Warning threshold, but below the Critical threshold, the resource is in a concern state. When resource usage is at or above the Critical threshold, the resource is in a critical state.

   You can set thresholds for the following system resources:

   **Memory usage**
   Measures system memory usage in GBs.

   Default Warning threshold - 70 percent of installed system memory, expressed in GBs.

   Default Critical threshold - 80 percent of installed system memory, expressed in GBs.

   **CPU usage**
   Measures CPU usage as a percentage of total CPU capacity on the system.

   Default Warning threshold - 70

   Default Critical threshold - 95

   **Disk usage**
   Measures disk usage as a percentage of all system disk storage. The default **Concern at** range for Disk Usage is 65 to 85.

   Default Warning threshold - 65

   Default Critical threshold - 85

7. If you want to be alerted by email when a resource reaches a critical state:
   a) Select the **Alert at critical condition** option for the resource.
   b) Enter the email addresses of the users to receive notification in the **Notify email ID(s)** box. When typing, you can separate addresses with a comma, space, tab, or enter. You can also paste a comma-delimited list of email addresses in this box.

      For Planning Analytics Administration on Cloud, notification emails are sent from administration@planning-analytics.ibmcloud.com.

For Planning Analytics Administration Local, notification emails are sent from the mail address specified by the PAA_EMAIL_ADDRESS property in the `bootstrap.properties` file for a given agent. The default value for this property is administration@planning-analytics.ibmcloud.com, but you can modify it to use an email address that is more relevant for your environment. See "Install and configure the Planning Analytics Administration agent (local only)" on page 919 for details on the `bootstrap.properties` file.

c) Click **Apply**.

8. You can click **Restore to original values** at any time to restore the default threshold and alert settings. Note that **Restore** also removes any email addresses from the **Notify email ID(s)** box.

# Install and configure the Planning Analytics Administration agent (local only)

If you have IBM Planning Analytics Local version 2.0.5 or later installed, you can install and configure the Planning Analytics Administration agent for Microsoft Windows or Linux operating systems.

To use IBM Planning Analytics Administration on Planning Analytics Workspace Local, you must install and configure the Planning Analytics Administration agent wherever you have installed IBM TM1 Server. The default port of the Planning Analytics Administration agent is 9012.

By default, the Planning Analytics Administration agent is selected as a component when you install Planning Analytics Local but it is not configured or started.

**Note:** You can upgrade your Planning Analytics Administration agent by installing a new version of Planning Analytics Local or by downloading a new version of the agent from Planning Analytics Workspace. For more information, see Planning Analytics Administration agent in Download additional components.

To upgrade an Planning Analytics Administration agent as part of a Planning Analytics Local upgrade, you must stop the Planning Analytics Administration agent service, back up your `bootstrap.properties` file, upgrade Planning Analytics Local selecting the Planning Analytics Administration agent component in the installation wizard, and then restart the Planning Analytics Administration agent. For more information, see Upgrading Planning Analytics Local.

Your changes to the `bootstrap.propertes` file are preserved when you upgrade the Planning Analytics Administration agent, however, the version of the agent is updated. On Windows, the version is updated when you upgrade using the Planning Analytics Local installation wizard. On Linux, the version is updated when you run `./startup_agent.sh install`.

After you install or upgrade the agent, you must configure it. For more information, see "Configure the agent for Windows" on page 919 or "Configure the agent for Linux" on page 920.

## Configure the agent for Windows

### Procedure

1. Open the Windows Services desktop application.
2. Stop the **IBM Planning Analytics Administration Agent** service if it is running.
3. Navigate to *<PA_install_location>*`/paa_agent/bin`.
4. Run `PAAAgentSetJavaHome.bat` to set JAVA_HOME for your Planning Analytics Administration agent.
5. Start the **IBM Planning Analytics Administration Agent** Windows service.

   **Note:** You can also navigate to *<PA_install_location>*`/paa_agent/bin` and run the `PAAAgentStart.bat` script to start the Planning Analytics Administration agent Windows service.

### What to do next

Navigate to Planning Analytics Administration in Planning Analytics Workspace Local and verify that the TM1 databases appear.

**Note:** If you change the `bootstrap.properties` file later, you must restart the **IBM Planning Analytics Administration Agent** Windows service by running `PAAAgentStart.bat` script.

## Configure the agent for Linux

### Before you begin

Set the RunningInBackground parameter in your `tms1.cfg` file to `RunningInBackground=T`.

**Note:** The `RunningInBackground` parameter is required for Linux only. It is used to suppress the prompts displayed by TM1 Server. Since Planning Analytics Administration uses scripts to start and stop databases on TM1 Server, you must set `RunningInBackground=T` on Linux.

### Procedure

1. Navigate to *<PA_install_location>*`/paa_agent/bin`.
2. Run the `./shutdown_agent.sh` command to stop the **IBM Planning Analytics Administration Agent** if it is running.
3. Navigate to *<PA_install_location>*`/paa_agent/wlp/usr/servers/kate-agent`.
4. In a text editor, open the `bootstrap.properties` file.
5. Set the full path of the directory that contains TM1 databases data directories to SERVERS_DIR.

   **Note:** Multiple paths must be separated by a semicolon. For example, `/opt/ibm/cognos/tm1_64/samples/tm1/;/srv/prod/servers/`.
6. Save and close the `bootstrap.properties` file.
7. Navigate to *<PA_install_location>*`/paa_agent/bin`.
8. Run `./set_java_home.sh <Full path to JRE>` to set JAVA_HOME for your Planning Analytics Administration agent.
9. Run `./startup_agent.sh install` to set up the Planning Analytics Administration agent service.

   **Note:** You must have root or sudo privileges to perform this step.
10. Run `./startup_agent.sh` command to start the Planning Analytics Administration agent.

### What to do next

Navigate to Planning Analytics Administration in Planning Analytics Workspace Local and verify that the TM1 databases appear.

**Note:** If you change the `bootstrap.properties` file later, you must restart the **IBM Planning Analytics Administration Agent** by running `./startup_agent.sh` command.

## Set an agent API key

When a request is sent from Planning Analytics Workspace to a local agent, an authentication token is also sent. This token is created using a key that must be set to the same value in both the `bootstrap.properties` file for the local agent and Planning Analytics Workspace Local configuration file.

### Procedure

1. Open the Windows Services desktop application.
2. Stop the **IBM Planning Analytics Administration Agent** service if it is running.
3. Open the `bootstrap.properties` file for the local agent.

The default location of `bootstrap.properties` is `C:\Program Files\ibm\cognos\tm1_64\paa_agent\wlp\usr\servers\kate-agent\`.

4. Add the **AGENT_API_KEY** property to `bootstrap.properties` and set the key property value to any plain text string of your choice.

   **Important:** Take note of the string you enter. You'll need it when you add the **AdminAgentAPIKey** parameter to the Planning Analytics Workspace Local configuration file in Step 8.
5. Save and close the `bootstrap.properties` file.
6. Restart the **IBM Planning Analytics Administration Agent** service.

   After the agent starts, the key property value you entered is encoded. The **AGENT_API_KEY** property is removed from `bootstrap.properties` and replaced with the **AGENT_API_KEY_ENCODED** property, which is set to the encoded version of the key property value you entered.

   If you want to change the key value, you must again add the **AGENT_API_KEY** property to the `bootstrap.properties` file. The **AGENT_API_KEY_ENCODED** property will then be updated to contain the encoded key.
7. Open the Planning Analytics Workspace Local configuration file.

   The configuration file is located in the `<planning_analytics_install_dir>\config` directory and is named `paw.ps1` (Windows) or `paw.env` (Linux).
8. Add the **AdminAgentAPIKey** parameter to the Planning Analytics Workspace Local configuration file and set the parameter value to match the **AGENT_API_KEY** plain text property value that you entered in Step 4.
9. Save the configuration file.
10. Run `./scripts/paw.ps1` (Windows) or `./scripts/paw.sh` to apply the configuration changes.

## Configure the Planning Analytics Administration Agent with an existing keystore

You can replace the default keystore used by Planning Analytics Administration agent with your own.

### Before you begin

The TM1 Admin Server and TM1 Server must already be secured using custom certificates/keystore

Back up the `<pa_install_dir>\paa_agent\wlp\usr\servers\kate-agent\server.xml` file to a different directory.

#### *Set the PA_INSTALL_DIR environment variable*

### Procedure

1. Open a Command Prompt as administrator.
2. Execute this command to set the environment variable:

   ```
   set PA_INSTALL_DIR=<path_to_Planning_Analytics_installation_dir>
   ```

   For example, `PA_INSTALL_DIR=C:\Program Files\ibm\tm1_64`.

#### *Convert the IBMTM1 keystore to PKCS12 format for Planning Analytics Administration agent*

### Procedure

1. Open Windows Services panel.
2. Stop the **IBM Planning Analytics Administration Agent** service.
3. Open a Command Prompt as administrator.

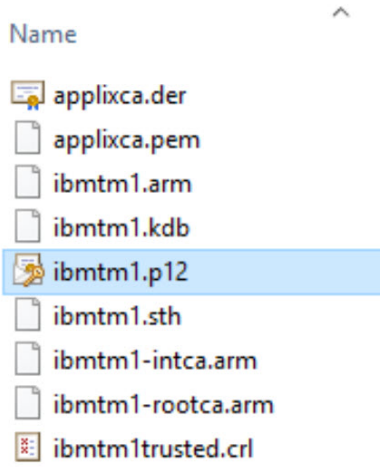
4. Execute this command to navigate to the \bin64 directory in the Planning Analytics installation directory:

   ```
   cd "%PA_INSTALL_DIR%\bin64\"
   ```

5. Execute this command to convert the ibmtm1 keystore file to a PKCS12 keystore for the Planning Analytics Administration Agent:

   ```
   gsk8capicmd_64 -keydb -convert -db "%PA_INSTALL_DIR%\bin64\ssl\ibmtm1.kdb" -stashed
   -old_format kdb
   -new_db "%PA_INSTALL_DIR%\bin64\ssl\ibmtm1.p12" -new_pw "CustomPA!@" -new_format pkcs12
   ```

6. Review the <Planning_Analytics_installation_dir>\bin64\ssl\ directory to confirm that the ibmtm1.p12 file is present.




7. Open the <Planning_Analytics_installation_dir>\paa_agent\wlp\usr\servers\kate-agent\server.xml file.
8. Update the **keyStore** to point to the new ibmtm1.p12 custom keystore:

   ```
   <keyStore id="defaultKeyStore" location="../../../../../bin64/ssl/
   ibmtm1.p12" type="pkcs12" password="CustomPA!@"/>
   ```

9. Save and close server.xml.

### What to do next

Any applications that communicate with the Planning Analytics Administration Agent must be updated to trust the new custom certificates For example, Configure TLS between Planning Analytics Workspace Local and other servers.

## Configure event notifications

### About this task

To get notifications from Planning Analytics Administration for Planning Analytics Local, you must configure the following functionality:

### Procedure

1. Set the following bootstrap properties for SMTP notifications:

   For example, set these properties:

   ```
   SMTP_EMAIL_PORT=587
   SMTP_EMAIL_AUTH=true
   SMTP_EMAIL_HOST=example.com
   SMTP_EMAIL_USERNAME=user@example.com
   ```

```
SMTP_EMAIL_PASSWORD=Analytics123
PAA_EMAIL_ADDRESS=noreply@example.com
```

The PAA_EMAIL_ADDRESS must be a registered alias, otherwise set it to a primary alias, for example, the user name noreply@example.com.

2. Optional: Set SMTP_EMAIL_START_TLS_ENABLE=true and add the certificates to the TLS certificate store:
   a) Run the following command in a command prompt on Windows or a terminal on Linux.

   ```
   openssl s_client -showcerts -starttls smtp -crlf -connect example.com:587
   ```

   This command prints out the certificates used by the SMTP server. Each certificate begins with "-----BEGIN CERTIFICATE-----" and ends with "-----END CERTIFICATE-----".

   b) Copy these two certificates (including the BEGIN CERTIFICATE and END CERTIFICATE lines) to Example_1.cer and Example_2.cer and remember the location.

   For example, <PA install directory>/paa_agent/wlp/usr/servers/kate-agent/resources/security. You use this location in the next step.

   c) Import Example_1.cer to the Planning Analytics Administration agent store by using the following command and the location of the Example_1.cer file from step "2.b" on page 923.

   On Windows:

   ```
   keytool.exe -importcert -keystore "<PA install directory>/paa_agent/wlp/usr/servers/kate-agent/resources/security/server_store.p12" -storetype PKCS12
   -trustcacerts -storepass applix -file "<PA install directory>/paa_agent/wlp/usr/servers/kate-agent/resources/security/Example_1.cer" -alias example1
   ```

   On Linux:

   ```
   ./keytool -import -keystore "<PA install directory>/paa_agent/wlp/usr/servers/kate-agent/resources/security/server_store.p12" -storetype PKCS12
   -trustcacerts -storepass applix -file "<PA install directory>/paa_agent/wlp/usr/servers/kate-agent/resources/security/Example.cer" -alias example1
   ```

   d) Repeat the previous step to import the second certificate, Example_2.cer.

### What to do next

You can get notifications for threads that are in a run state, threads in a wait state, and more. For more information, see Set system resource thresholds and alerts in the *Planning Analytics Workspace* documentation.

## Enable email notifications

You can configure a connection to your SMTP or SendGrid email services so that you can share assets such as books, views, images, and links with others.

To enable the email service, specify the following parameters in the Planning Analytics Workspace Local configuration file:

1. Set ENABLE_EMAIL to true to enable email service in Planning Analytics Workspace Local.
2. Use the EMAIL_SMTP_URL or SENDGRID_API_KEY parameter to configure settings.

   If you specify both EMAIL_SMTP_URL and SENDGRID_API_KEY parameters, only EMAIL_SMTP_URL is used.

   **EMAIL_SMTP_URL**

   Use the EMAIL_SMTP_URL parameter to enable the SMTP email service. The default value of this parameter is empty. The following are examples of SMTP URLs:

   - EMAIL_SMTP_URL="smtp://username:password@smtp.example.com:587"

- `EMAIL_SMTP_URL="smtp://username:password@smtp.example.com:587?secure=false"`
- `EMAIL_SMTP_URL="smtps://username:password@smtp.example.com:465"`

Where, `username` and `password` are your credentials and `smtp.example.com` is the server (for example, `smtp.office365.com`).

**Note:** If the port number of your SMTP server is 465, the SMTP URL starts with `"smtps://"`. Port 465 is typically the secure port for SMTP servers. For port number 587, use `"smtp://"` in the SMTP URL. Port 587 is a non-TLS port connection to an SMTP server.

**SENDGRID_API_KEY**
Use this parameter to enable the SendGrid email service; default is empty. This parameter specifies the API key for the SendGrid email account.

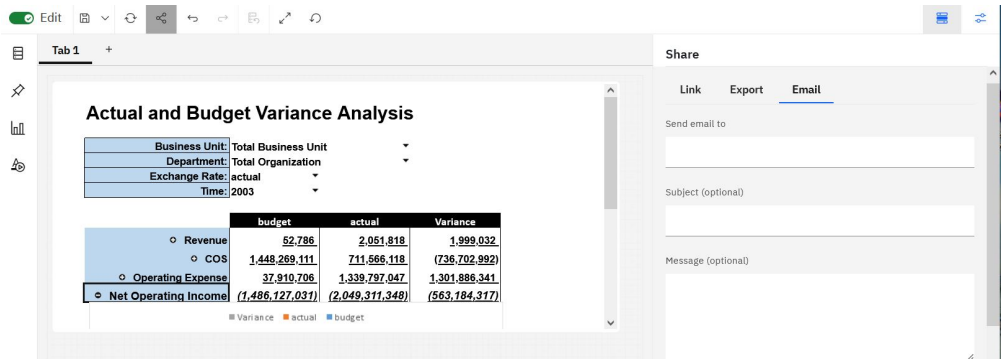
You might need to contact the email administrator for your organization to obtain the API key that is specific to your email service.

**Note:** If you use `SENDGRID_API_KEY`, you need to also specify the `EMAIL_FROM` parameter.

3. Use `EMAIL_FROM` to specify the sender's email address. This parameter might be required by the email server and is mandatory when you use `SENDGRID_API_KEY`.
4. If your SMTP server uses a custom TLS certificate, add the certificate to the `paw/config/certs` folder.
5. Restart Planning Analytics Workspace services.

**Note:** When you change a paw configuration file, you must run `./scripts/paw` for your operating system or click **Start** in the Planning Analytics Workspace administration tool for your changes to take effect. Only services that are affected by the configuration change will restart.

The **Email** tab is now available in the user interface when you share a book or view.




## Sample bootstrap.properties file

```
# General properties
SERVERS_DIR=<PA_install_location>/samples/tm1
EXE_PATH=<PA_install_location>/bin64/tm1s.exe
SERVER_INFO_PATH=<PA_install_location>/paa_agent/paaAgentCache
AGENT_LABEL=<"agent name">

TM1_SERVER_INFO_PROCESS_TIME_INTERVAL=20000
NOTIFICATION_MEM_USAGE_INTERVAL=30000
NOTIFICATION_SERVER_UNRESPONSIVE_INTERVAL=40000
FILE_TAILER_REFRESH_INTERVAL=86400000
PAA_EMAIL_ADDRESS=administration@planning-analytics.ibmcloud.com
MODEL_UPLOAD=model_upload
SCRIPT_TIMEOUT=15

# SMTP Properties
SMTP_EMAIL_PORT=<port>
SMTP_EMAIL_AUTH=false
SMTP_EMAIL_HOST=<host>
SMTP_EMAIL_START_TLS_ENABLE=false
SMTP_EMAIL_USERNAME=<username>
SMTP_EMAIL_PASSWORD=<encrypted_pwd>
```

```
# Logging properties
com.ibm.ws.logging.console.log.level=OFF
com.ibm.ws.logging.trace.specification="*\=audit\:com.ibm.pa.kate.agent.*\=warning"
com.ibm.ws.logging.max.file.size=10
com.ibm.ws.logging.max.files=4

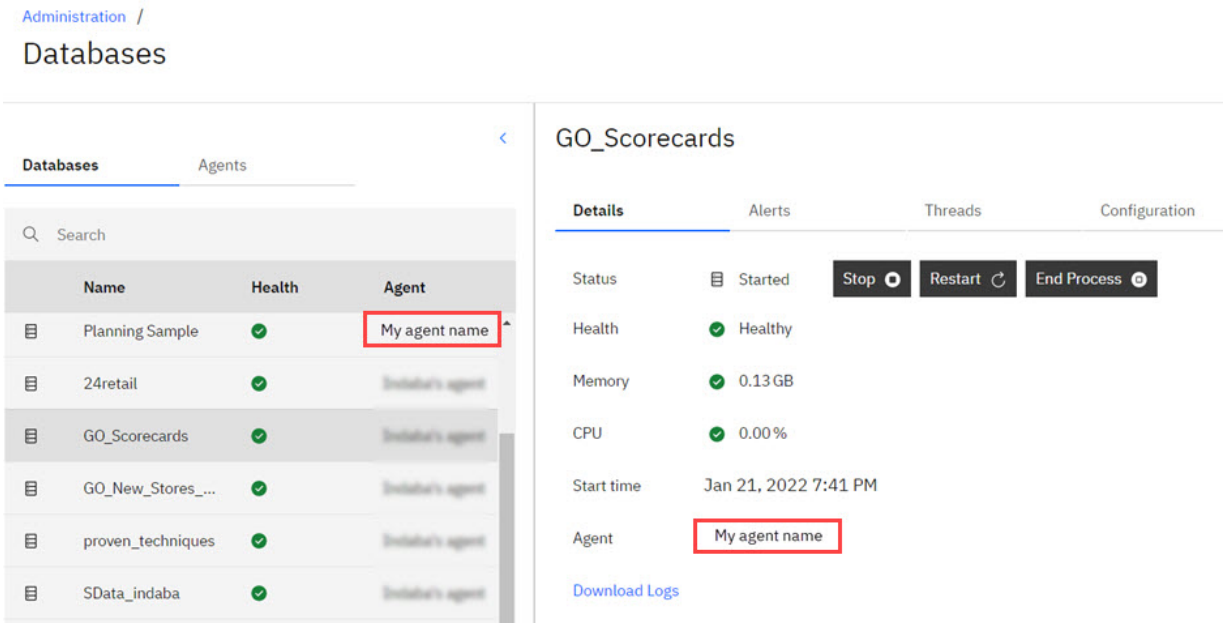
VERSION=1.0.nn.nnn
```

## General properties

**AGENT_LABEL**

Use this setting to display a specified agent label (name) in place of the agent IP address in Planning Analytics Administration.

If an agent label is set, the label appears on the **Databases** and **Agent** tabs wherever an agent is identified. The agent IP address is not reported. For example:




If an **AGENT_LABEL** is not specified, the agent IP address is displayed wherever an agent is identified.

**EXE_PATH**

Internal use only. Do not change.

**FILE_TAILER_REFRESH_INTERVAL**

Internal use only. Do not change.

Time in milliseconds between successive task executions. Used in scheduler that tails server logs for every TM1 Server.

**MODEL_UPLOAD**

Internal use only. Do not change.

**NOTIFICATION_MEM_USAGE_INTERVAL**

Internal use only. Do not change.

Not currently used.

**NOTIFICATION_SERVER_UNRESPONSIVE_INTERVAL**

Internal use only. Do not change.

Time in milliseconds between successive task executions. Used in the scheduler that checks for server unresponsiveness.

**PAA_EMAIL_ADDRESS**

Internal use only. Do not change.

**SCRIPT_TIMEOUT**

Internal use only. Do not change.

**SERVERS_DIR**
This parameter is applicable to Linux only. It specifies the full path of the directory that contains the TM1 databases directories.

**Note:**

Multiple paths must be separated by a semicolon. For example,

```
SERVERS_DIR=C:/tm1/samples/tm1/;C:/prod/servers/
```

You can specify the parent directory of your databases. You don't need to specify each database separately.

**SERVER_INFO_PATH**
Internal use only. Do not change.

**TM1_SERVER_INFO_PROCESS_TIME_INTERVAL**
Internal use only. Do not change.

Time in milliseconds between successive task executions. Used in the scheduler that checks system health.

## SMTP properties

**SMTP_EMAIL_PORT**
Set to a free port. For example, `2500`.

**SMTP_EMAIL_AUTH**
Default is `false`.

**SMTP_EMAIL_HOST**
Set based on your email server. For example, `localhost`.

**SMTP_EMAIL_START_TLS_ENABLE**
Default is `false`. Whether you need to configure certificates and set START_TLS_ENABLE=`true` depends on your email server configuration.

**SMTP_EMAIL_USERNAME**
Full email address. For example, example@my domain.com.

**SMTP_EMAIL_PASSWORD**
Email password for SMTP_EMAIL_USERNAME.

## Logging properties

**com.ibm.ws.logging.console.log.level**
This setting controls the granularity of messages that go to the console.

When this log is enabled, a `console.log` is generated with the agent `message.log` at *<PA_install_location>*`/paa_agent/wlp/usr/servers/kate-agent/logs`.

Valid values are: INFO, AUDIT, WARNING, ERROR, and OFF.

Default is OFF.

**com.ibm.ws.logging.trace.specification="*=audit:com.ibm.pa.kate.agent.*=warning"**
This setting is used to selectively enable trace.

This entry refers to the agent `message.log`.

Valid values are: INFO, AUDIT, WARNING, ERROR, and OFF.

A restart of the Planning Analytics Administration agent is required for a logging value to take effect when you change this property.

**com.ibm.ws.logging.max.file.size**
The maximum size (in MB) that a log file can reach before it is rolled.

Default is 10. For more information, see Logging and Trace.

**com.ibm.ws.logging.max.files**

If an enforced maximum file size exists, this setting is used to determine how many of each of the logs files are kept.

Default is 4. For more information, see Logging and Trace.

# Monitor and manage database threads

You can view a detailed report of thread activity on any running database. Using the information available in the report, you can cancel threads or disconnect users that are negatively affecting database performance.

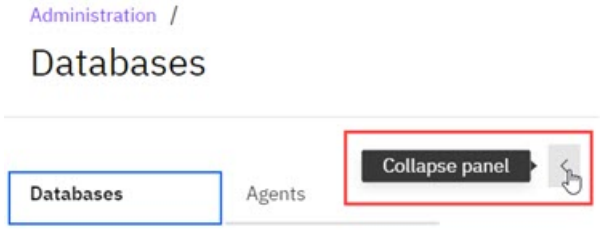
## Monitor and manage database thread activity in Planning Analytics Workspace Local and Planning Analytics Workspace on Cloud

### About this task

**Note:** This procedure applies only to Planning Analytics Workspace Local and Planning Analytics Workspace on Cloud. For details on similar capabilities in Planning Analytics Workspace on Cloud Pak for Data or Planning Analytics Workspace on Planning Analytics as a Service, see “Additional administration capabilities for IBM Planning Analytics as a Service” on page 955

To open a database thread report, click the **Databases** tab on the Databases page, then click the database for which you want to view a report. Next, click the **Threads** tab to display the report.

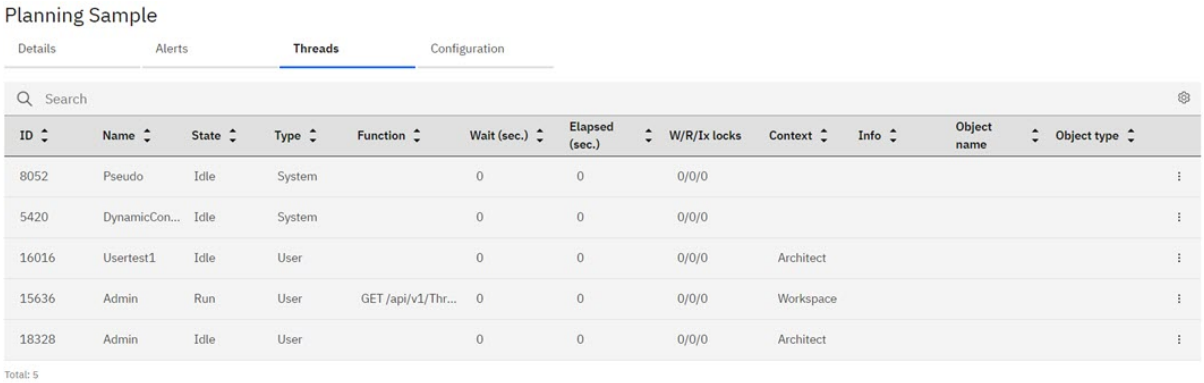
**Tip:** To view the full report, you might have to collapse the panel on the **Databases** page.




When you open the **Databases** page as an administrator, you can see a list of all databases running in your environment. When you attempt to open a database thread report, Planning Analytics attempts to authenticate you as an administrator on the selected database using the same username and password you used to log into Planning Analytics. If your administrator credentials on the selected database are different from those you used to log into Planning Analytics, authentication will fail and you'll be prompted to provide a valid administrator username and password for the database.

A database thread report provides details on every thread on your database, in table format. You can't change the column widths in the report, but you can hover over any item in the report to see the full value.

You can search for multiple words on the Threads tab. Separate each word with a comma. For example, `run,admin,Architect`. Search is not case-sensitive.



For each thread you can view these details:

**ID**
The unique numeric ID of the thread.

**Name**
The name of the user who initiated the thread.

**State**
The current thread state.

- Idle: not using resources
- Wait: waiting on other threads, blocked
- Running
- Commit: committing updates made by a thread
- Rollback: rolling back to a state before an update
- Login: when a user logs in and initiates a thread

**Type**
Indicates a User or System thread.

**Function**
The API function being executed by the thread.

**Wait**
The length of time, in seconds, that the thread has been in a wait state.

**Elapsed**
The total elapsed time of the thread in its current state.

**W/R/Ix locks**
The number of write, read, and intent-to-write locks that the thread holds.

A write lock allows only the thread that holds the lock to access and write changes to an object. No other thread can read or modify the object until the W lock has been released.

A read lock allows many threads to read from an object at the same time, but does not allow another thread to modify or write to the object until all R locks have been released.

An intent-to-write lock reserves the right for a thread to obtain a write lock on an object when all read locks have been released. Only one thread at a time is allowed to have an Ix lock on an object.

**Context**
The connected Planning Analytics client that is associated with the thread.

**Info**

When a thread is in a wait state, this column displays the ThreadID of the thread that is blocking the waiting thread.

**Object name**
The name of the object upon which the thread is operating.

**Object type**
The type of object upon which the thread is operating.

You can click ⚙ to selectively hide or show any of these columns in your report. The column display configuration is retained between sessions. The next time you open a thread report for any database, you'll see the same columns.

## Procedure

1. To sort values by any column, click directly on a column name to cycle through the sort options.
2. To cancel a thread, click the **Thread actions** button ⋮, then click **Cancel thread**.

You can cancel any user thread, including threads you own. You cannot cancel system threads.

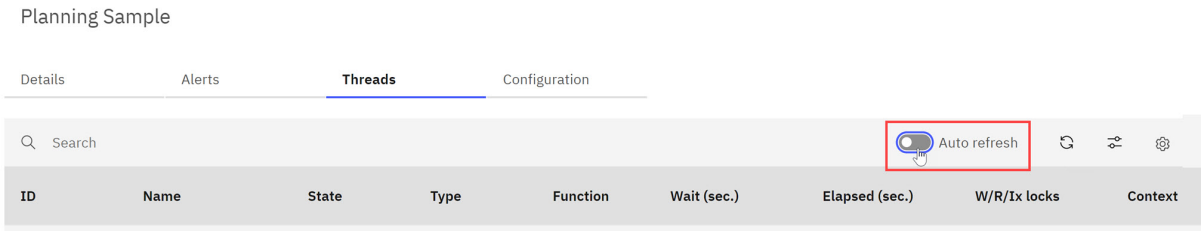
3. To disconnect a user, click the **Thread actions** button ⋮, then click **Disconnect user**.
   You can disconnect user threads, but not system threads. You cannot disconnect yourself.

### Temporarily pause thread or activity refresh while analyzing a database

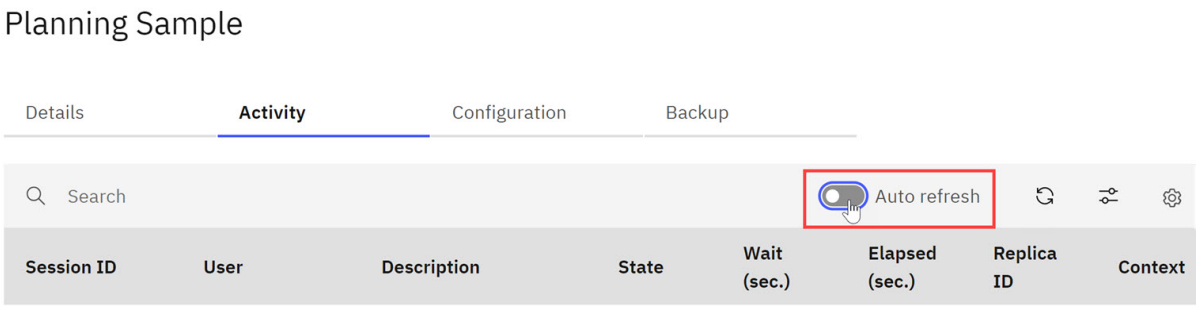
The **Auto refresh** option on the database page lets you control automatic thread or activity refresh. You can use this option to pause refresh when you want to examine database activity without being interrupted by updates.

The option is available in Planning Analytics Workspace on all platforms, but is located slightly differently according to platform.

- In Planning Analytics Workspace Local and Planning Analytics Workspace on Cloud, the option is available on the database **Threads** tab.




- In Planning Analytics Workspace on Cloud Pak for Data and Planning Analytics Workspace on Planning Analytics as a Service, the option is available on the database **Activity** tab.




To temporarily pause automatic refresh, click the **Thread** or **Activity** tab, then toggle the **Auto refresh** option to the off position.

Thread or activity collection continues while automatic refresh is paused. You can click **Refresh** to manually refresh the thread or activity list while automatic refresh is paused.

When you toggle the **Auto refresh** option to the off position, the setting does not persist when you leave the **Thread** or **Activity** tab. If you click a different tab or a different database, the **Auto refresh** option is reset to the on position.

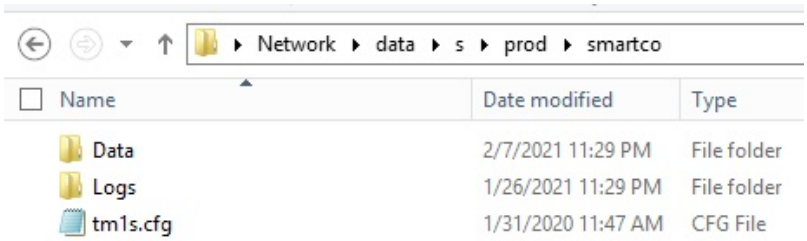
## Create a database (Cloud only)

Planning Analytics on Cloud users can create their own Planning Analytics databases in Planning Analytics Administration.

All databases are created in the `prod` folder. The default database is named TM1.

If the `prod` folder doesn't exist, it is created along with the `Data` and `Logs` folders, and the `tm1s.cfg` file. You can use the folder as is, or change the directory structure to organize the Planning Analytics objects in a specific way. By default, all objects reside in the base directory. If the `prod` folder with valid subfolders and the `tm1s.cfg` file already exists, the folder is used for the new database.

The best practice is to keep the folder structure as shown in the following figure:




In this example, the name of the application is `smartco`. Therefore, for this Planning Analytics application, the base directory is: `\\data\s\prod\smartco`.

The following conditions must be met when creating and using the Planning Analytics database directory:

- The `tm1s.cfg` file must exist in the base directory.
- Any folders that you create must be within the base directory.
- The location or the name of the base directory cannot be changed.

Typically, customers create additional folders with the following names:

- `Excel` - contains Perspectives report and template definitions for TM1 Web.
- `Files` - used to transfer `.txt` data files.

The list of settings in the tm1s.cfg file that you should not change is the same as for other Planning Analytics databases. If you need to change any settings, use Planning Analytics Workspace Administration to ensure that you change the proper the settings.

All users with the Administrator role in Planning Analytics Workspace are given ADMIN access to the new Planning Analytics database.

The default security mode 1 "admin" account for the new Planning Analytics database has a randomly generated password. This password cannot be accessed by the Cloud Operations team, and therefore cannot be communicated to you through the Welcome Kit.

If access to the default security mode 1 "admin" account is required, a new password can be set by using the AssignClientPassword function using your IBMid administrator account.

If access control lists are enabled to control access to shared folders, the access controls are applied to the database as well. For more information, see access control lists (the sections about user access to shared folders, and TM1 database access to data in shared folders).

## About this task

When you specify the Planning Analytics database name, ensure that you follow rules regarding special characters, character lengths, and other operating system and process restrictions. The database name cannot:

- Have more than 75 characters
- Be blank
- Contain only spaces, and begin or end with a space
- Contain only periods (for example, '...'), and end with a period '.'
- Contain only the number 0
- Contain the following characters: \/":|<>+=;,?*@$ ` '}{%
- Contain the new line (\n) character

**Note:** Planning Analytics database names are available publicly on the TM1 Web login screen if allow lists for web services are not applied. Choose a database name that does not expose sensitive information or implement allow lists for web services.

### Procedure

1. Log on to Planning Analytics.
2. In the Welcome page, click **Administration**.
3. Click **Databases**.
4. On the **Databases** tab, click **Add**.

   The **New database** dialog box is displayed.
5. In the **Database name** field, enter the name for your database.

   Ensure that the name complies with the restrictions that are specified earlier in this topic.
6. Click **Create**.

   While the process runs in the background (asynchronously), you can work on other tasks, including adding another Planning Analytics database. The process of creating multiple databases is performed in a sequential order, and not in parallel. Planning Analytics Workspace Administration continues to check the database creation process, and displays a message when the process is complete.

   The database creation process, even for one database, might take a few minutes to complete, especially if access control lists to shared folders are applied. The time is even longer if you added multiple databases.

   The new database or databases are added to the list of databases on the **Databases** tab.
7. Locate your newly added database name or names in the list of databases. To sort names in the alphabetical order, click the **Name** heading.

### What to do next

You can proceed to configure the database settings. For more information, see “Configure databases” on page 934.

To delete a Planning Analytics database, use the self-service delete function in Planning Analytics Workspace Administration. For more information, see “Delete a database” on page 932.

## Rename a database

Planning Analytics on Cloud users can rename databases by using Planning Analytics Administration.

The rename process affects the database in the following ways:

- The database name is changed in the user interface.
- The database directory in the `prod` directory is renamed with the new name.

  If the directory rename cannot succeed, the whole process is canceled, any actions are reverted, and an error message is presented to the user.
- The database name in the Planning Analytics Workspace configuration is updated to match the new name.
- Any ACL files provided by customers are updated in the product repository if it includes the folder that was renamed.

### Before you begin

You must open a support request to rename a Cognos Controller Financial Analytics Publisher (FAP) databases. You cannot rename a FAP databases using Planning Analytics Administration.

### About this task

A database must be stopped before you can rename it. Plan your renaming for a time when you can stop the database if necessary.

When you specify the new database name, follow the same rules regarding special characters, character lengths, and other restrictions that are documented in the topic “Create a database (Cloud only)” on page 929.

### Procedure

1. Log in to Planning Analytics Workspace as a member with the Administrator role.
2. Click the **Administration** tile.
3. Click the **Databases** tile.
4. Click the database that you want to rename on the **Databases** pane.
5. On the database details pane, check the database status. If the status is **Started**, click **Stop**.

   The database must be stopped before it can be renamed. It might take a few minutes to stop the database. When the process is complete, the database icon changes to stopped  .
6. Click Edit on the database details pane.
7. Enter a new name for the database, then click the **Save** .

   The rename process might take several minutes to complete. When it's finished, the new database name is displayed on the **Databases** pane.

### What to do next

Any existing content, such as rules or processes, that references the database name must be updated to refer to the new database name.

## Delete a database

Planning Analytics on Cloud users can delete TM1 databases by using Planning Analytics Workspace Administration.

The system imposes the following restrictions on this functionality:

- Last TM1 database cannot be deleted.

  At least one database must always exist in Planning Analytics Workspace Administration.
- The database `tm1s.cfg` file is missing or contains errors.

  For example, the `ServerName` parameter is missing in the file.

  The errors are not shown in Planning Analytics Workspace Administration, and are most likely caused by manual modifications to the `tm1s.cfg` file. For more information, see the topic Can I change the tm1s.cfg file?
- Cognos Controller Financial Analytics Publisher (FAP) databases cannot be deleted.

### Before you begin

The ability to delete a TM1 database is not available in the Planning Analytics Workspace Classic Experience. For information about enabling the New Experience, see this article.

The database server must be stopped before you can delete the database.

### About this task

Deleting the database in the user interface doesn’t delete the database directory and its contents from the data tier. The database directory, and the folders and files in it, are left for you to decide if they should be deleted, moved, copied, or kept on the data tier. It is strongly recommended to move the directory elsewhere if you don't intend to recreate the database.

**Note:** To recreate a deleted database, you can create a new database with the same name. The database will use the existing directory and its contents if they are still on the data tier.

### Procedure

1. Log on to Planning Analytics as a member of the Administrator role.
2. In the Welcome page, select **Administration**.
3. Click **Databases**.
4. In the **Databases** panel, locate the database that you want to delete, and click its name.
5. On the **Details** tab, check the database status. If the status is **Started**, click **Stop**.

   The database must be stopped before it can be deleted. It might take a few minutes to stop the database. When the process is complete, the database icon changes to stopped  .

   You can now proceed to delete the database.
6. In the **Databases** panel, select the database that you want to delete.
7. In the toolbar, click the delete icon  .

   A dialog box with a message is displayed.
8. In the dialog box, select the **Delete this database** checkbox, and then click the **Delete** button.

   The database is deleted from the user interface, but the database directory is retained.

### Results

The database name no longer appears in the **Databases** panel.

Also, the database is no longer available for use in Planning Analytics Workspace Books, as well as Planning Analytics for Microsoft Excel Explorations and Reports. Any existing content that leverages the database will now have errors.

### What to do next

Decide what to do with the database directory and its contents. You can copy, move, or delete the directory, or keep in on the data tier. For more information about files and folders in the database directory, see "Create a database (Cloud only)" on page 929.

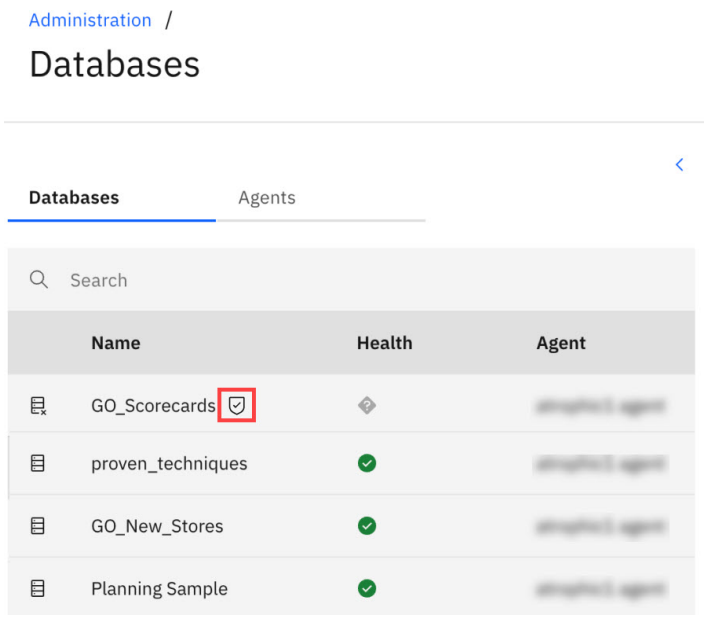
## Identify the database used for authentication on Planning Analytics Administration Local (Local only)

You can quickly identify the database used for authentication in the list of databases on the Planning Analytics Administration page.

**Important:** For the authentication database to be correctly identified in Planning Analytics Administration, the TM1LoginServerURL configuration parameter for Planning Analytics Workspace Local must be specified with an IP address and port. The IP address must match the address specified by the IPAddressV4 parameter in your `tm1s.cfg` file. The port number must match the port specified by the HttpPortNumber parameter in your `tm1s.cfg` file.

```
TM1LoginServerURL=https://
IPAddressV4_from_tm1s.cfg:HttpPortNumber_from_tm1s.cfg
```

The database used for authentication is identified by the  icon. The icon is displayed only on Planning Analytics Administration Local and only when Planning Analytics Workspace is configured to authenticate against a specific Planning Analytics database.




Administrators can look for this icon to determine which database is used for authentication so they don't accidentally stop the database or modify it in such a way that prevents users from being able to log in to Planning Analytics Workspace.

# Configure databases

You can set configuration parameters for your databases, set the thread auto-refresh interval for the database activity report, and configure system resource thresholds and alerts.

## Set database configuration parameters

You can set configuration parameters that determine the behavior of your databases.

### About this task

The Planning Analytics Administration interface provides access to Planning Analytics database parameters. Only the parameters listed here can be set through Planning Analytics Administration.

Parameters are organized in **Administration**, **Modelling**, **Performance**, and **Access** categories, and then further arranged by functional category.

The Planning Analytics Administration interface provides a brief description for each parameter, along with an indication of the default parameter value.

**Administration category**
**Default**

AllowReadOnlyChoreReschedule

DisableSandboxing

Language

MaskUserNameInServerTools

PerfMonActive

PerformanceMonitorOn

RunningInBackground

StartupChores

UnicodeUpperLowerCase

**Audit log**
Enable

**Clients**
ClientPropertiesSyncInterval

**External databases**
SQLRowsetSize

**Server log**
Enable

ExcelWebPublishEnabled

**Top log**
Enable

TopScanFrequency

**Modelling category**
**Default**
DefaultMeasuresDimension

EnableNewHierarchyCreation

MDXSelectCalculatedMemberInputs

UserDefinedCalculations

**Rules**
AllowSeparateNandCRules

AutomaticallyAddCubeDependencies

ForceReevaluationOfFeedersForFedCellsOnDataChange

RulesOverwriteCellsOnLoad

**Spreading**
ProportionSpreadToZeroCells

SpreadingPrecision

**Startup**
LoadPrivateSubsetsOnStartup

MaximumCubeLoadThreads

PersistentFeeders

StartupChores

**Synchronization**
MaximumSynchAttempts

**TI**
EnableTIDebugging

MaximumTILockObjects

UseExcelSerialDate

**Performance category**
**Default**
PrivilegeGenerationOptimization

**JobQueuing**
Enable

MaxWaitTime

ThreadPoolSize

ThreadSleepTime

**Locking**
PullInvalidationSubsets

UseLocalCopiesForPublicDynamicSubsets

**Memory**

ApplyMaximumViewSizeToEntireTransaction

CacheFriendlyMalloc

DisableMemoryCache

LockPagesInMemory

MaximumMemoryForSubsetUndoKB

MaximumViewSizeMB

MaximumGlobalQueryMemorySizeMB

MaximumSingleQueryMemorySizeMB

**MTCubeLoad**

Enabled

MinFileSize

UseBookmarkFiles

Weight

**MTFeeders**

Enabled

AtStartup

**MTQ (multi-threaded queries)**

MTQQuery

MTQ.ImmediateCheckForSplit

NumberOfThreadsToUse

SingleCellConsolidation

**Stargate**

AllRuleCalcStargateOptimization

UseStargateForRules

ZeroWeightOptimization

**View calculation**

CalculationThresholdForStorage

CheckFeedersMaximumCells

MagnitudeDifferenceToBeZero

ViewConsolidationOptimization

**Access category**

**Authentication**

MaximumLoginAttempts

**CAM**

CreateNewCAMClients

**CAPI**

MessageCompression

ProgressMessage

**HTTP**

SessionTimeout

## Procedure

1. On the Databases page, click the **Databases** tab.
2. In the list of databases, click the database for which you want to set configuration parameters.

3. Click the **Configuration** tab.
4. Locate the parameter that you want to modify, then set a new parameter value.
5. Click **Apply**.

   Database configuration parameters are identified on the Configuration tab as either dynamic or static. Dynamic parameter values can be changed without requiring a database restart. Changes to static parameter values require a database restart.
6. You can click **Restore to original values** at any time to restore all of the listed parameters to their default values.

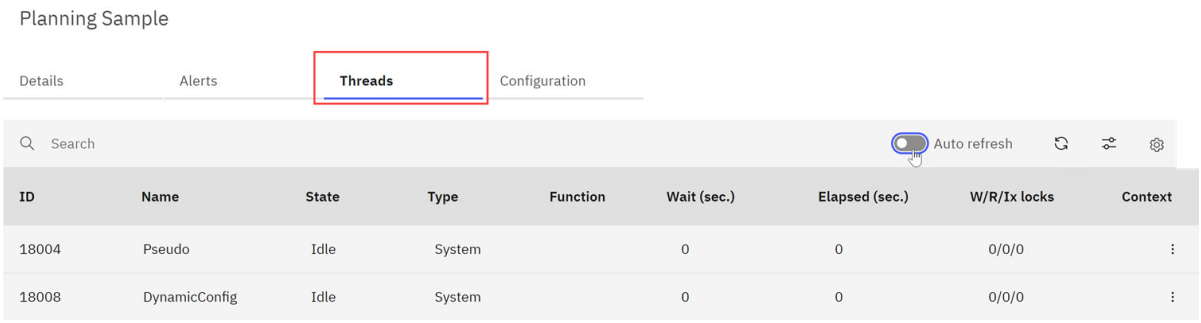
## Set the thread or activity auto-refresh interval

By default, thread or activity status on the database report is updated every 50 seconds. You can modify the refresh interval if you want to refresh thread or activity status more frequently.

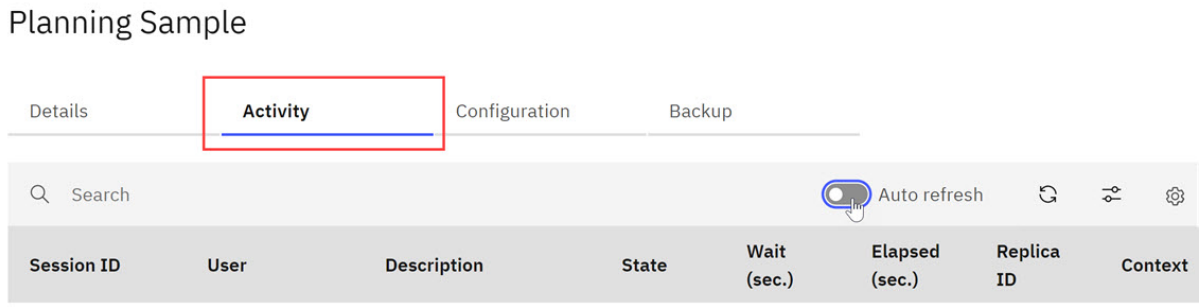
### *Set the thread or activity auto-refresh interval Planning Analytics Workspace*

### About this task

In Planning Analytics Workspace Local and Planning Analytics Workspace on Cloud, the refresh interval applies to the database **Threads** tab.




In Planning Analytics Workspace on Cloud Pak for Data and Planning Analytics Workspace on Planning Analytics as a Service, the refresh interval applies to the database **Activity** tab.




### Procedure

1. Click a database on the **Databases** administration page.
2. Click the **Threads** or **Activity** tab.
3. Click
4. Use the slider to set a different refresh interval

## Set database thresholds and alerts

You can configure settings that define database status by multiple metrics. You can also configure alerts to send notification to yourself and other users when any of the metrics reaches a critical state.

### About this task

When you set threshold limits, you define the parameters for the safe, concern and critical states of these metrics for an individual database:

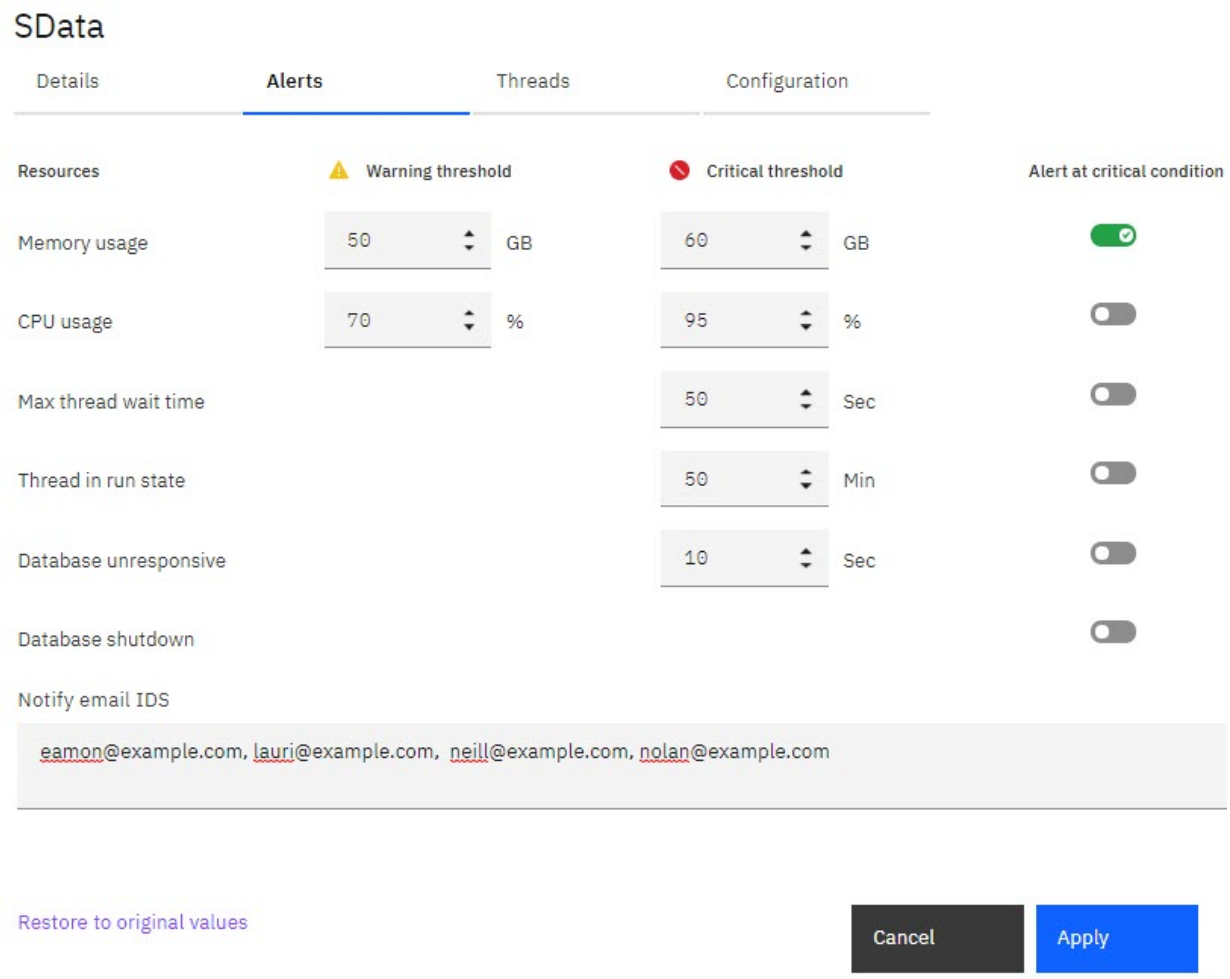
- Memory usage
- CPU usage
- Maximum thread wait time
- Duration of thread in run state
- Database unresponsive

**Note:** If you're using TM1 Database 12, you can set alerts and thresholds for Memory usage and CPU usage metrics only.

### *Set database thresholds and alerts in Planning Analytics Workspace*

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace Home page.
2. Click **Databases**.
3. On the **Databases** tab, click the database for which you want to set thresholds and alerts.
4. Click **Alerts**.




5. For the following database metrics, set the **Warning threshold** and **Critical threshold**.

   When the metric value falls below the Warning threshold, the database is in a healthy state. When the metric is at or above the Warning threshold, but below the Critical threshold, the database is in a concern state. When the metric is at or above the Critical threshold, the database is in a critical state.

   **Memory usage**
   Measures database memory usage in GBs.

   **CPU usage**
   Measures database CPU usage as a percentage of total CPU capacity on the system.

6. For the following database metrics, set the **Critical threshold**.

   When the metric value is at or above the Critical threshold, the database is in a critical state. Otherwise, the database is in a safe state.

   **Max thread wait time**
   Measures the amount of time a threads waits to run, expressed in seconds

   **Thread in run state**
   Measures the duration of a thread in a run state, expressed in minutes

   **Database unresponsive**
   Measures the duration of a database in an unresponsive state, expressed in seconds.

7. If you want to be alerted by email when a database metric reaches a critical state or when the database is shut down:

   a) Select the **Alert at critical condition** option for the metric or for database shutdown.

b) Enter the email addresses of the users to receive notification in the **Notify email ID(s)** box. When typing, you can separate addresses with a comma, space, tab, or enter. You can also paste a comma-delimited list of email addresses in this box.

   For Planning Analytics Administration on Cloud, notification emails are sent from administration@planning-analytics.ibmcloud.com.

   For Planning Analytics Administration Local, notification emails are sent from the mail address specified by the PAA_EMAIL_ADDRESS property in the `bootstrap.properties` file for a given agent. The default value for this property is administration@planning-analytics.ibmcloud.com, but you can modify it to use an email address that is more relevant for your environment. See "Install and configure the Planning Analytics Administration agent (local only)" on page 919 for details on the `bootstrap.properties` file.

c) Click **Apply**.

8. You can click **Restore to original values** at any time to restore the default threshold and alert settings. **Restore** also removes any email addresses from the **Notify email ID(s)** box.

# Start and stop databases

You can start, stop, or restart databases from the **Databases** administration page.

## About this task

On Planning Analytics Workspace Local, you cannot use the Databases page to stop, start, or restart a database that is running as an application. The database must be running as a service if you want to manage the database from the Planning Analytics Monitoring dashboard.

## Start and stop databases in Planning Analytics Workspace

### Procedure

1. On the **Databases** administration page, click the **Databases** tab, then click the database you want to work with.
2. Click the **Details** tab for the database.
3. Click **Start** to start a database that is stopped.
4. Click **Stop** to stop a database that is running normally. Click **Apply** when you're asked for confirmation.
5. Click **Restart** to stop and then restart a database that is running normally. Click **Apply** when you're asked for confirmation.
6. If a database enters an unresponsive or deadlocked state, you can click **End Process** to end the database process. Click **Apply** when you're asked for confirmation.

   **Note: Do not** use this option to stop a database that is running normally. The **End Process** option should be reserved only for databases in an unresponsive state. The **End process** option kills all the running threads in the database. Any data that is not logged prior to ending the database process is lost.

## Automatically start a database when the data tier is restarted (Cloud only)

The **Details** tab on the database administration page includes an **Auto-start** option to automatically start a database when the data tier is restarted during regular monthly maintenance of Planning Analytics on Cloud.

### About this task

It's possible that you don't want some of your databases to start when the data tier is restarted, while other databases are essential and must be started along with the data tier. You can use the **Auto-start** option to control the behavior of individual databases. By automatically starting only essential databases upon data tier restart, you can conserve system resources and reduce database startup time.

The **Auto-start** option is available only on Planning Analytics Administration on Cloud, it is not available on Planning Analytics Administration Local.

### Procedure

1. On the Databases administration page, click the **Databases** tab, then click the database you want to work with.
2. Click the **Details** tab for the database.
3. To automatically start the database when the data tier restarts, enable the **Auto-start** option.
4. To require a manual restart of the database when the data tier restarts, disable the **Auto-start** option.

## Download database log files

You can download several types of logs for a database. See Planning Analytics and TM1 logs for descriptions of the log files.

### Download database log files in Planning Analytics Workspace

### Procedure

1. On the **Databases** page, click the database for which you want to download logs.
2. On the database Details tab, click **Download Logs**.
3. On the **Download Logs** page, select the log files you want to download.

   To make it easier to find the log files you need, you can click a column heading to sort by **Version** (file name), **Date**, or **Size**. You can also search for files.

   If a log file is locked by a process, selection of the locked file is disabled and the log file cannot be downloaded.

   **Note:** You can download up to 500 MB of log files. If the files you select to download total more than 500 MB, the **Download** button is disabled. You must clear selections until the total of files is less than 500 MB.

   Additionally, if the logging directory for your database contains more than 3000 files, Planning Analytics Workspace is unable retrieve the list of files and will throw an error. To resolve this issue, archive or delete old log files until there are less than 3000 files in the logging directory.
4. Click **Download Logs**.

   The log files are saved in a .zip archive using the database name, `<database_name>tm1serverLogs.zip` The .zip archive is saved in your `Downloads` directory.

## Save data for a selected database

To save all data for a database from memory to disc, right-click the database in the Data tree, then click **Save data**. This action also restarts the `tm1s.log` file for the database.

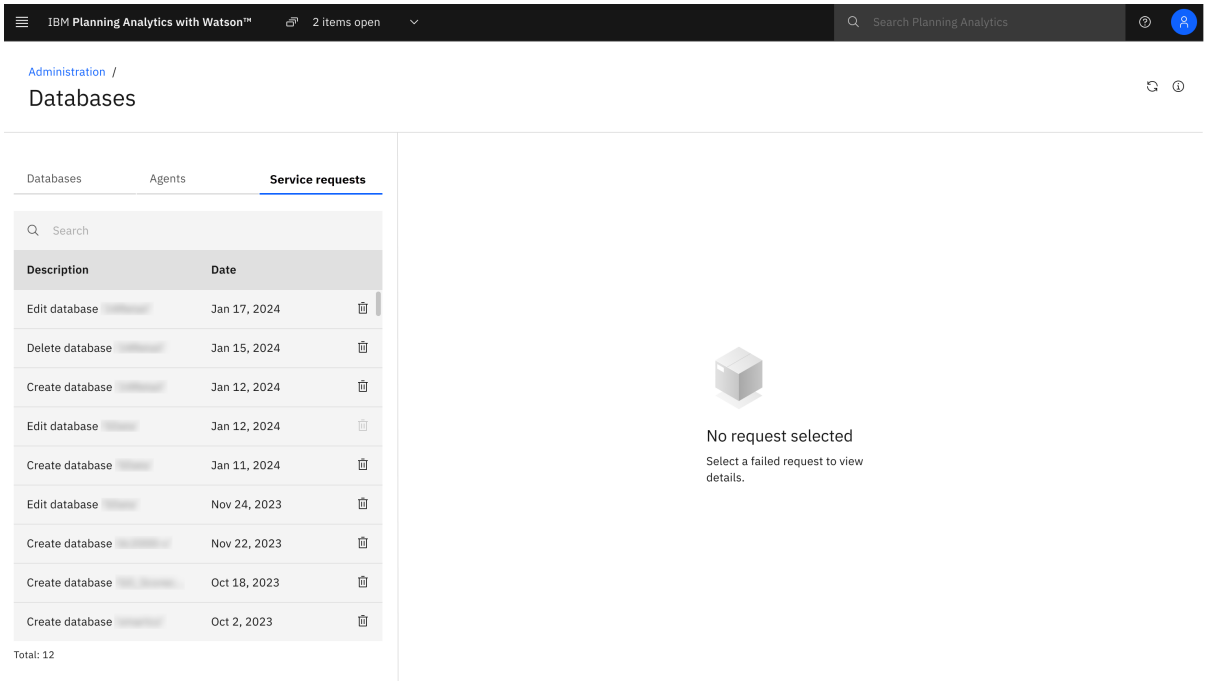
## Refresh security for a selected database

To update all security structures and assignments for a database, right-click the database in the Data tree, then click **Refresh security**.

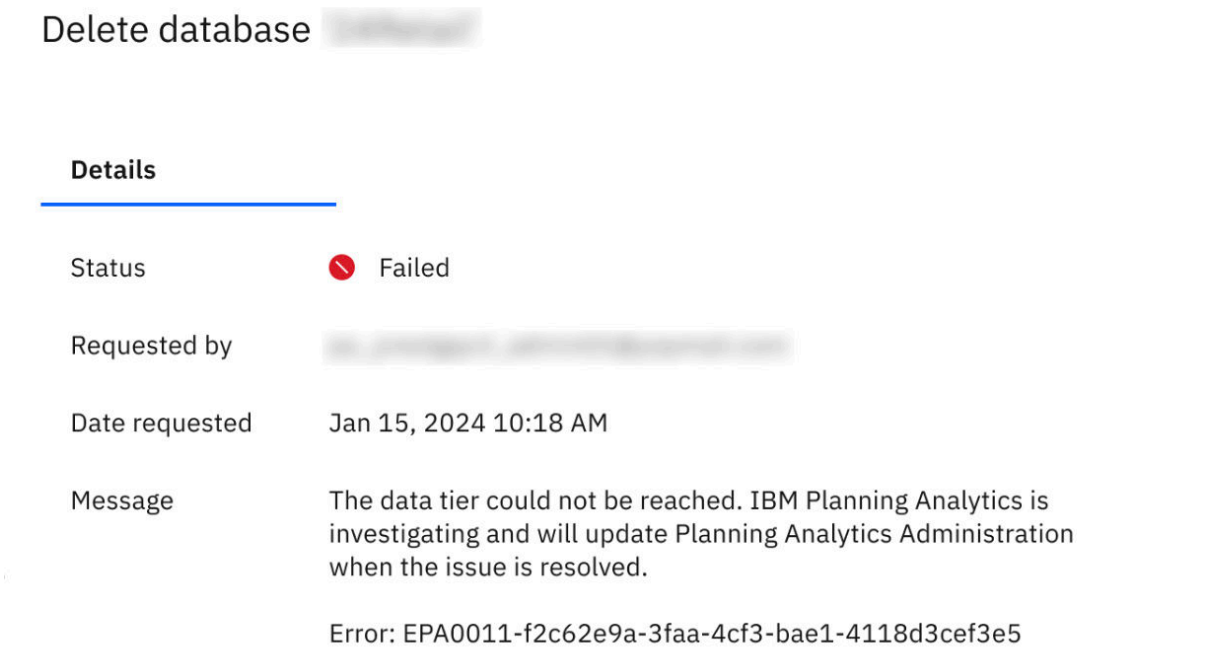
## View failed database requests

You view failed self-service database requests in IBM Planning Analytics Administration.

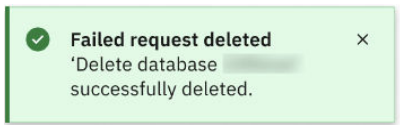
A tab named **Service requests** is available on the **Databases** page. This tab contains a list of failed self-service database requests. You can sort the **Service requests** list by description or by the date the requests were created.




When you click a failed request, the **Details** panel displays information about the failed request, including the status of the request, who made the request, when the request was made, and why the request failed.




You can click the delete icon to delete a request on the **Service requests** tab. A notification appears when the request has been successfully deleted.



# Download additional components

You can download the following components of IBM Planning Analytics directly from Planning Analytics Administration.

## Planning Analytics for Microsoft Excel (cloud only)

 To download Planning Analytics for Microsoft Excel, follow these steps.

### Procedure

1. On the Planning Analytics Workspace Home page, click the **Administration** tile.
2. On the Administration page, click **Excel and Customizations**.
3. Select the version of Planning Analytics for Microsoft Excel you want to download. The three most recent versions are available for selection.
4. Click **Download Update**.

   The installation is saved as an archive named Integration<version>.zip in your Downloads directory.
5. Extract the archive.
6. Consult the `README.txt` file in the extracted archive for installation instructions.

   You can also refer to Downloading and upgrading to Planning Analytics for Microsoft Excel with a single .xll add-in for versions 2.0.65 and later for installation instructions.

## Planning Analytics Administration agent (local only)

To upgrade the Planning Analytics Administration agent, follow these steps.

### Before you begin

- You must have IBM Planning Analytics Local version 2.0.5 or later installed wherever you have installed IBM TM1 Server.
- You must ensure that port 9012 is available. The default port of the Planning Analytics Administration agent is 9012.
- You must install and configure the Planning Analytics Administration agent. For more information, see “Install and configure the Planning Analytics Administration agent (local only)” on page 919.

### *Download the Planning Analytics Administration agent in Planning Analytics Workspace*

### Procedure

1. On the Planning Analytics Workspace Home page, click the **Administration** tile.
2. On the Administration page, click **Agents**.
3. Click the **Download** button  , then click **Download Agent**.

   The installation is saved as an archive named `paa-agent-pkg-<version>.zip` in your Downloads directory.

### *Update the Planning Analytics Administration agent*

### Procedure

1. Extract the archive to a temporary location. .
2. Open a command window and change to the directory where you extracted the compressed folder.

**Note:** You will not be able to stop a service or modify files in <*PA_install_location*>\paa_agent\ unless you have the required permissions. On Windows, open the command window with `Run as administrator`.

**Tip:** Put the full path to <*PA_install_location*> in quotation marks if it contains spaces.

a) To update the Planning Analytics Administration agent Windows service, run **UpdatePAAAgent.bat**, passing in the full path to <*PA_install_location*>, that is, the parent of the `paa_agent` directory.

b) To update the Planning Analytics Administration agent service on Linux, run **UpdatePAAAgent.sh**, passing in the full path to <*PA_install_location*>, that is, the parent of the `paa_agent` directory.

The **UpdatePAAAgent** script stops the Planning Analytics Administration agent before updating it.

Your existing `bootstrap.properties` file is preserved and updated with the new agent version number. For example, `VERSION=1.0.39.736`.

**Note:** The **UpdatePAAAgent** script restarts the Planning Analytics Administration agent at the end of the script.

# Customize the interface for your company

Administrators can upload fonts, color palettes, and themes to be used in IBM Planning Analytics Workspace.

## Upload custom fonts

Administrators can upload custom fonts that can be used in views and text boxes.

You can upload the following file types: .tff (True Type), .otf (OpenType), .woff (Web Open Font Format), .eot (Embedded OpenType), .svg (Scalable Vector Graphics), and .woff2 (Web Open Font 2.0 Format).

**Note:** Not all font types are supported by all browsers. The font type works only if it is supported by the browser.

The font file naming convention is:

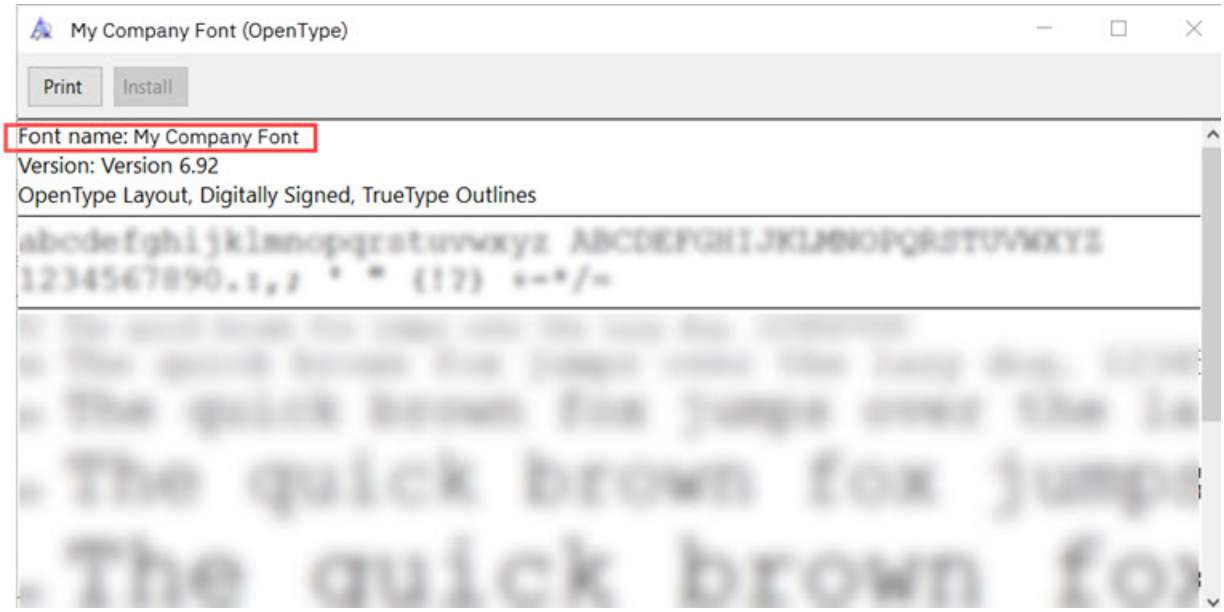
<*family name*>-<*style identifiers*>.<*extension*>

For example:

- `Exotic Custom Font-Thin.ttf`
- `Exotic Custom Font-BoldItalic.ttf`

This file name formatting enables font families to be grouped in the text properties.

Starting with Planning Analytics Workspace 2.0.75, custom fonts can be displayed in websheets. To successfully display a custom font that has been uploaded to Planning Analytics Workspace in websheets, the font file name must exactly match the font name as defined in the font. For example, this font name is defined as `My Company Font`.




The font file must be named `My Company Font.otf` to make the font available in websheets.

### Uploaing custom fonts in Planning Analytics Workspace

#### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace Home page.
2. Click the **Excel and Customizations** tile.
3. Click the **Fonts** tab.
4. Click the **Upload** icon.
5. Either drag the font file onto the **Drag your font file here** box, or click **Tap to browse**, and select the font file.
6. Close any opened books and then reopen them.

#### Results

The font is now available for selection in **Text Properties**.

## Create a global color palette

Administrators can create global color palettes that can be applied in books and visualizations in Planning Analytics Workspace.

### About this task

You can create the following types of color palettes:

**Categorical**
Used for visualizations that support discrete colors, like a bar or pie chart.

**Continuous**
Used for visualizations that support color transitions, like a map or a heat map.

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace Home page.
2. Click the **Excel and Customizations** tile.

3. Click the **Palettes** tab.
4. Click **Add global palette** +.

   The **Create color palette** window opens. The palette type is set to categorical by default. To create a continuous palette, click the **Continuous palette** icon. To switch to a categorical palette, click the **Categorical palette** icon.
5. Enter a name for your palette.
6. Click the **Grid** or **Wheel** tab.

   In the **Grid** tab, you can select colors from a grid of color swatches. In the **Wheel** tab, you can select a color by doing one of the following things:

   - Click the color wheel
   - Enter the color value in HSB (hue, saturation, brightness) or RGB (red, green, blue) notation
   - Enter the color value in hex code
   - Use the sliders or arrow buttons to change HSB or RGB values
7. Under **Color guide**, click **Automatic** or **Custom**.

   **Categorical color palette**

   Contains a set of individual color swatches.

   In **Automatic**, when you select a color from the grid or wheel, all swatches in the palette are filled with colors that are related to the color you selected, starting from the currently selected swatch. In **Custom**, you must select each swatch and then select a color for it.

   **Continuous color palette**

   Contains one continuous swatch. **Automatic** is not available for a continuous color palette.

   In **Custom**, a continuous swatch initially contains a start point and an end point. When you select a color from the grid or wheel, the swatch is filled with the color you selected. The color gradually fades in intensity from the start point to the end point of the swatch.

   Alternatively, you can click the start point and select one color, then click the end point and select another color. In this case, the swatch contains a continuous transition from the color at the start point to the color at the end point. For example, this swatch shows a continuous transition from yellow at the start point to red at the end point.

   
8. To add more swatches to the palette, click **Add swatch**.

   On a categorical palette, a new swatch is added to the end of the palette. Click the new swatch, then select a color.

   On a continuous palette, an interval point is inserted into the continuous swatch. The interval point is inserted so that the continuous swatch is equally divided between the start point, the end point, and any interval points. You can insert multiple interval points into a continuous swatch. To assign a color to an interval point, click the interval point then select a color.

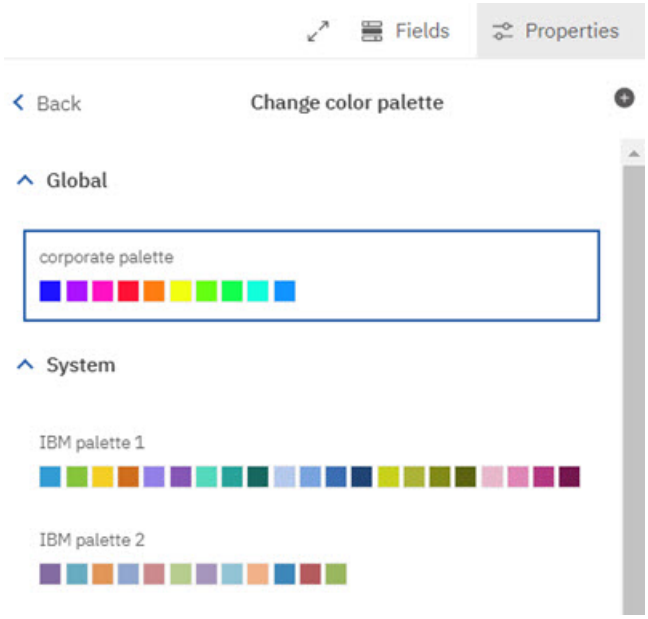
   This example shows a continuous swatch with two interval points between the start point and end point. It begins with green at the start point, than transitions between yellow and red interval points, and concludes with black at the end point.

9. To undo a selection, click **Remove swatch** .
10. To reverse the colors in the palette, click **Reverse palette** .
11. Click **Save** when you are finished.

### Results

The color palette is available for use under **Global** color palette properties.




If an administrator modifies a global color palette, the changes are applied anywhere the palette is used.

If an administrator deletes a global color palette, any items that use the global palette revert to the system default palette.

Color palettes can be migrated between different environments using Lifecycle Management. If a book containing a color palette is migrated to a new environment, the color palette is added to the new environment once the book exports. Color palettes cannot be exported or imported separately. For more information, see Copy and deploy assets with Lifecycle Management in Planning Analytics Workspace.

## Create a custom theme

You can override the standard Planning Analytics Workspace theme with a custom theme that reflects your corporate branding. You can use a custom theme to modify the colors of the application bar and the navigation bar, set the corporate title that is shown on the home page. and specify an image to display on the home page.

### About this task

A theme is defined in a `.zip` archive that contains a `spec.json` file and an `images` folder.

The `spec.json` file contains the branding instructions for a theme, most notably color definitions and image paths. The `images` folder contains the images that you want to display in the theme.

The total size of the `.zip` archive for a theme cannot exceed 32 MB, inclusive of any image files. The `.zip` archive name cannot include underscores ( _ ). Image file names cannot contain spaces.

If you are administering Planning Analytics Workspace Local with multiple environments, a new theme is created for the environment that is active when you initiate the theme creation or upload process.

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace home page.

2. Click the **Excel and Customizations** tile on Planning Analytics Workspace cloud or the **Customizations** tile on Planning Analytics Workspace local .
3. Click the **Themes** tab.
4. Select the **System default** theme, then click **Download**.

   The theme archive named `sample-theme.zip` is downloaded to your `Downloads` directory. You use this theme as a template for a custom theme that specifies colors, images, and titles that reflect your corporate branding.

   **Note:** If you have previously defined a custom theme and want to modify it, you can select the existing theme and click **Download**. In this case, the `.zip` archive for the existing theme is downloaded to your `Downloads` directory.
5. Extract the `.zip` archive to a location of your choice.

   After you extract the archive, you see two items: a `spec.json` file and an `images` folder.

   The `spec.json` file looks like this:

```
{
    "schemaVersion":"3.0",
    "name": "sample-theme",
    "brandText":"Sample Theme",
    "images":{
        "brandIcon": "sample-theme/images/brand.svg",
        "favicon": this key is currently unsupported
    },
    "uiShellTheme": "dark",
    "colors":{
        "appbarBackground":"#161616",
        "appbarForeground":"#fff",
        "appbarSelectLine":"#fff",
        "appbarPushButtonBackground":"#393939",
        "navbarBackground":"#fff",
        "navbarForeground":"#161616",
        "navbarSelectLine":"#0f62fe",
        "navbarPushButtonBackground":"#e0e0e0",
        "personalMenuBackground": "inherit"
    }
}
```

6. Add the image that you intend to reference in the `spec.json` file to the `images` folder. Images in themes can be .svg, .png, .jpg, or .jpeg format.
7. Modify the key values in the `spec.json` file to specify the text, image, and colors you want to use in your theme. Colors are specified using hexadecimal color values.

*Table 14. Theme objects*

| JSON key name | Definition |
|---|---|
| `name` | The name for the theme. This name is displayed on the **Themes** tab of the **Excel and Customizations** page in Planning Analytics Workspace Administration. You cannot use the name `System default`, which is reserved for the system default theme. |
| `brandText` | The branding text to display on the **Planning Analytics Workspace home** page. |
| `brandIcon` | The branding image to display on the **Planning Analytics Workspace home** page. This image is displayed immediately prior and near the `brandText`. |
| `favicon` | Not currently supported. |

| *Table 14. Theme objects (continued)* | |
|---|---|
| **JSON key name** | **Definition** |
| `uiShellTheme` | Determines the appearance of items in the **Home** menu ☰. Specify either `light` (carbon 10) or `dark`(carbon 100) background. |
| `appbarBackground` | Application bar background color. This color is applied to the entire application bar, both clickable and non-clickable regions. |
| `appbarForeground` | Application bar foreground color. This color is applied to text and icons on the application bar.<br>This setting is not applied to the Switcher widget on the application bar. The Switcher displays a list of open items. The text of the Switcher always appears White. |
| `appbarSelectLine` | Application bar selection line color. This color is applied to the border that surrounds a selected or clicked item on the application bar. |
| `appbarPushButtonBackground` | Application bar push-button background color. This color is displayed when you hover or click an item on the application bar. |
| `navbarBackground` | Navigation bar background color. This color is applied to the entire navigation bar, both clickable and non-clickable regions. |
| `navbarForeground` | Navigation bar foreground color. This color is applied to text and icons on the navigation bar. |
| `navbarSelectLine` | Navigation bar selection line color. This color is applied to the bar that appears on a selected navigation bar item. |
| `navbarPushButtonBackground` | Navigation bar push-button background color. This background color is displayed when you click on an item on the application bar. |
| `personalMenuBackground` | Specifies the background color of the **Personal** menu icon 👤.<br>Specify `inherit` to inherit the `navbarBackground` and `navbarPushButtonBackground` colors. |

See "Application of theme keys in Planning Analytics Workspace" on page 950 for a graphical depiction of the application of theme keys in the user interface.

8. Save your changes, then include both the modified `spec.json` file and the `images` folder in a new theme `.zip` archive.
9. Click **Add** on the Themes tab.
10. Drag the new theme `.zip` archive onto the **Add theme** box or click the link to navigate to the `.zip` archive on your file system.

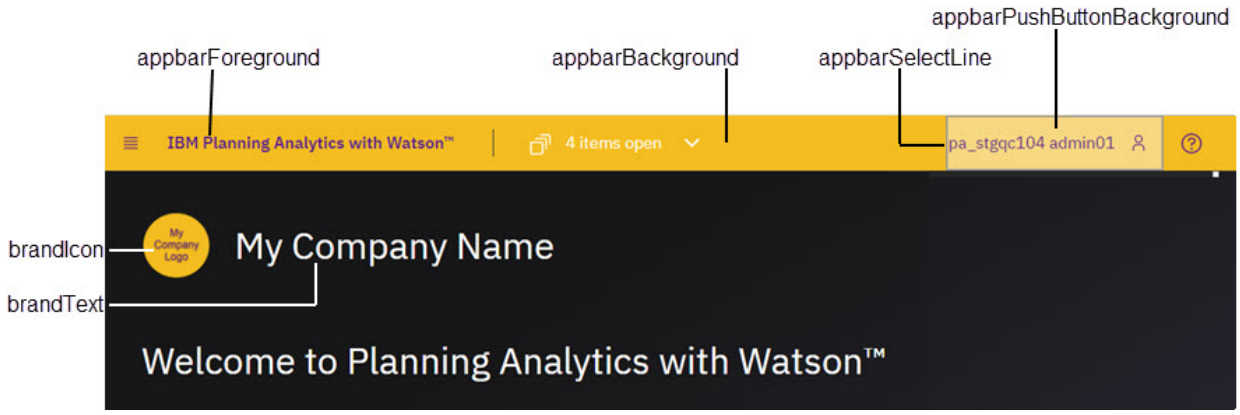
    The new theme is now available on the Themes tab. You can activate the theme to see your customizations in Planning Analytics Workspace.

## Application of theme keys in Planning Analytics Workspace

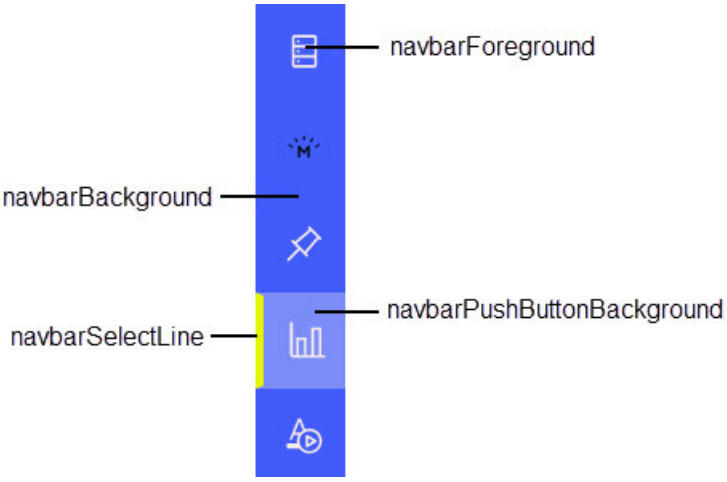
These images illustrate where each of the available `spec.json` keys are applied in Planning Analytics Workspace when you activate a custom theme.

For more information on setting these keys to customize your Planning Analytics Workspace interface, see "Create a custom theme" on page 947.

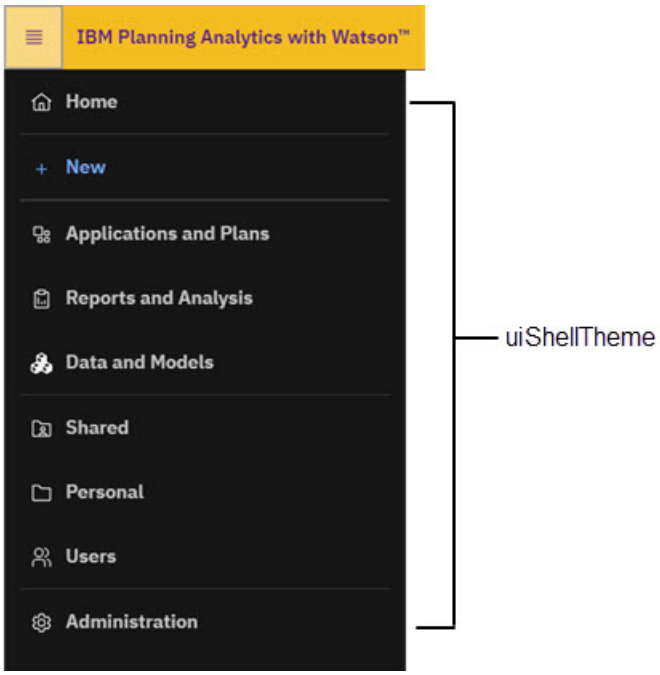
### Application bar and home page




### Navigation bar



### Home menu




## Activate a theme

After you create a theme, you must activate it to see the changes in Planning Analytics Workspace.

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace home page.
2. Click the **Excel and Customizations** tile on Planning Analytics Workspace cloud or the **Customizations** tile on Planning Analytics Workspace local .
3. Click the **Themes** tab.
4. Select the theme that you want to use in your user interface.
5. Click **Activate**.

   **Note:** As an administrator, you must refresh your browser to see the new theme settings in Planning Analytics Workspace after you activate a theme. Any user who logs in after a theme is activated immediately sees the theme; a browser refresh is not necessary.

## Delete a theme

You cannot delete the active theme or the System default theme.

### About this task

If you want to delete the active theme, you must first activate a different theme, then delete the previously active theme.

### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace Home page.
2. Click the **Excel and Customizations** tile.
3. Click the **Themes** tab.
4. Select the theme you want to delete.
5. Click .

6. Click **Delete** when you're asked to confirm the deletion.

### Update a theme

You cannot directly change an existing theme or upload changes to an existing theme.

#### About this task

To update a theme, you must first activate a different theme. You can then download the theme you want to modify, delete it from Planning Analytics Workspace, and then make any necessary modifications. Finally, you can upload the modified theme and activate it.

#### Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace home page.
2. Click the **Excel and Customizations** tile on Planning Analytics Workspace cloud or the **Customizations** tile on Planning Analytics Workspace local .
3. Click the **Themes** tab.
4. Activate a theme other than the one you want to update.
5. Select the theme that you want to update, then click **Download**.
6. Delete the theme you downloaded.
7. Update the theme and then upload the modified theme as described in “Create a custom theme” on page 947.
8. Activate the modified theme you uploaded. You might have to refresh your browser to see the theme in your Planning Analytics Workspace interface.

## Manage integrations

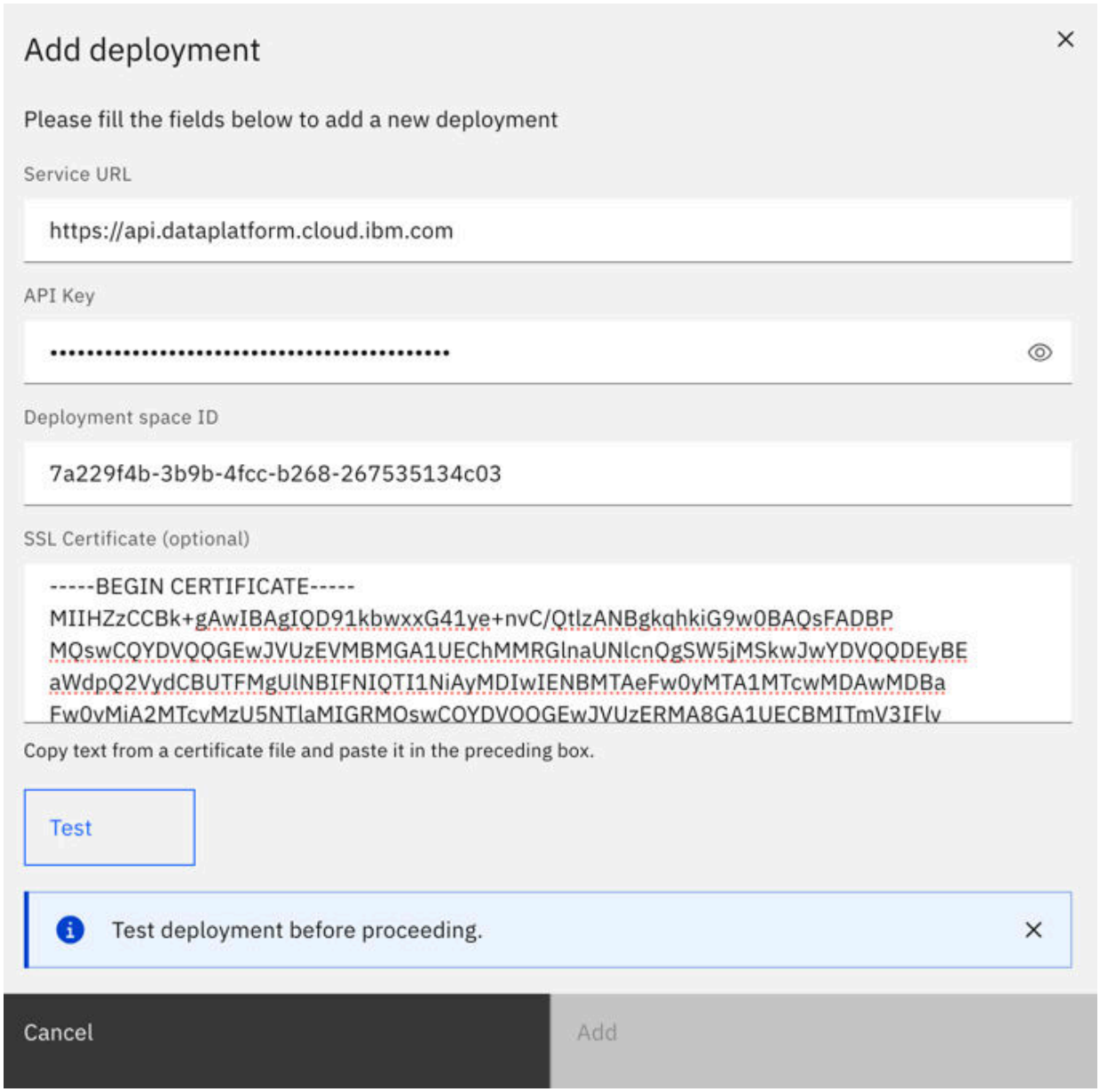
Use the **Integrations** tile to manage Planning Analytics Workspace integrations with other IBM products and services.

### Configure integration with IBM watsonx.ai to support Decision Optimization

You need to configure an integration with an existing Watson Machine Learning model on IBM watsonx.ai to enable Decision Optimization in Planning Analytics Workspace. This integration was previously called IBM Cloud Pak for Data Platform.

#### Procedure

1. On the Planning Analytics Administration page, click the **Integrations** tile.
2. **IBM watsonx.ai** displays in the list of available integrations. Click **Add** to add a new deployment.




3. Specify a **Service URL**.

   The Service URL identifies the Watson Data API endpoint to interact with to discover and execute Decision Optimization jobs. This URL varies based on your locale. Examples include: `https://api.dataplatform.cloud.ibm.com` and `https://api.eu-de.dataplatform.cloud.ibm.com`.

4. Specify an **API Key**.

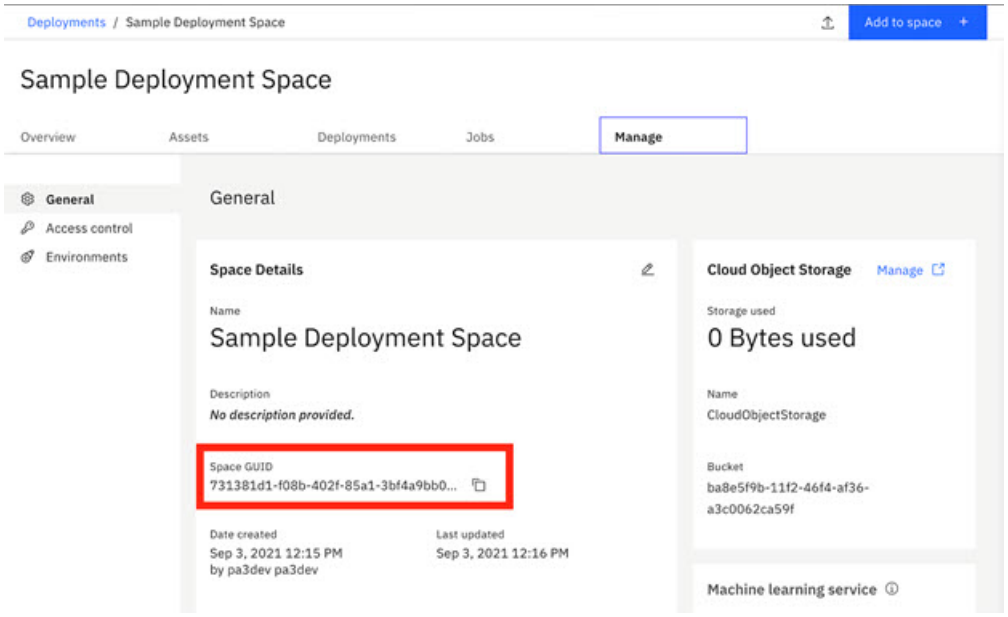
   The API Key, if you don't already have one, can be created in watsonx.ai through IBM Cloud Identity and Access Management (IAM).

   You can access IAM through the watsonx.ai menu, under **Administration** > **Access (IAM)**.

   The detailed steps to create this key are provided in the Creating an IAM bearer token section of the Watson Data API documentation.

5. Set the **Deployment Space ID**.

   The Deployment Space ID can be obtained from the **Space GUID** field of the **Manage** tab for the deployment space that contains the Decision Optimization jobs that you want to make available to your Planning Analytics users.




6. Paste the SSL certificate into the **SSL Certificate** box.

   The SSL Certificate (PEM) can be obtained from the browser. Most browsers have a lock icon beside the URL that you can click on to view the certificate. Be sure to retain the `-----BEGIN CERTIFICATE-----` and `-----END CERTIFICATE-----` portions of the certificate when you pasted the certificate into the **SSL Certificate** box.
7. Click **Test connection**.
8. Once the test is successful, click **Add** to finish adding the deployment.
9. Toggle the deployment status to **Active** to enable it.

### What to do next

You can configure multiple deployments of the IBM watsonx.ai integration. Click **Add** or use the context menu of an existing deployment to duplicate it and edit the relevant fields.

## Connect the Planning Analytics AI assistant to a custom instance of IBM watsonx Orchestrate

Use your own watsonx Orchestrate configuration to customize and extend the capabilities of the Planning Analytics AI assistant.

### About this task

Configure the Planning Analytics AI assistant to connect to a custom watsonx Orchestrate instance instead of the default instance. Using your own instance of watsonx Orchestrate extends the Planning Analytics AI assistant to start your own skills and intents (such as sending emails or triggering workflows) directly from the Planning Analytics AI assistant chat.

### Procedure

1. On the Planning Analytics Administration page, click the **Integrations** tile.
2. Click **IBM watsonx Orchestrate**.
3. Click **Custom instance**.
4. In the **Configuration value** field, paste your custom configuration JSON string.
5. Click **Test connection**.
6. If your connection is successful, click **Apply** to activate the configuration.

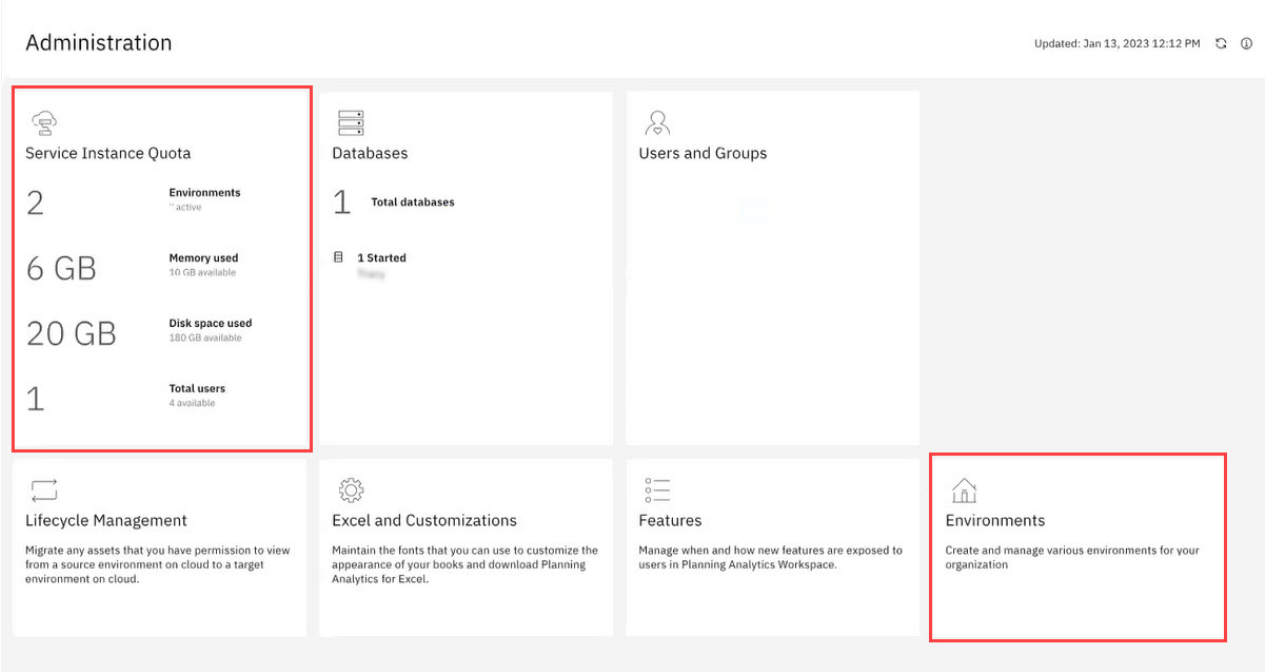
### Results

After you enable the custom configuration, you can interact with the Planning Analytics AI assistant as usual. For example, typing "`send an email`" triggers the corresponding AI activity from the connected custom watsonx Orchestrate instance if it is properly configured.

# Additional administration capabilities for IBM Planning Analytics as a Service

IBM Planning Analytics as a Service retains most of the administration capabilities of Planning Analytics Administration on Cloud, but there are some differences.

The following administration tiles are unique to Planning Analytics as a Service and provide capabilities not available in Planning Analytics Administration on Cloud.




The **Service Instance Quota** tile provides a read-only report of quota status for your Planning Analytics as a Service instance. This tile is not interactive; nothing happens if you click the tile.

The **Environments** tile lets you manage your IBM Planning Analytics as a Service environments.

In addition, the capabilities of the **Databases** tile are different on Planning Analytics as a Service than in Planning Analytics Administration on Cloud. For more information, see "Manage databases on Planning Analytics as a Service" on page 957.

## Review your Service Instance Quota for IBM Planning Analytics as a Service

The **Service Instance Quota** tile on the Planning Analytics Administration home page provides a read-only report of quota status for your Planning Analytics as a Service instance.

### Procedure

1. Open Planning Analytics Workspacein Planning Analytics as a Service.
2. Click the **Administration** tile.
3. Review the report on the **Service Instance Quota** tile.

   This report is not interactive, it just provides a quick overview of quota usage. Noting happens if you click the **Service Instance Quota** tile.

# Manage your IBM Planning Analytics as a Service environments

You can create or delete a Planning Analytics as a Service environment in Planning Analytics Administration.

## About this task

To create or manage environments, you need environment admin privileges. The first user added to Planning Analytics as a Service is given this privilege by default and it can be granted to other Administrators by enabling the toggle for that user.

Users can have different access privilege for each environment.

## Add an environment

## Procedure

1. Log in to your Planning Analytics as a Service instance.
2. On the Planning Analytics as a Service home page, click the **Administration** tile.
3. On the **Administration** page, click the **Environments** tab.
4. Click **Add** and enter the environment name.
5. Select the type of environment you want to create, **Development** or **Production**.
6. Click **Create**.

   The new environment is available on the environment selector on the IBM Planning Analytics title bar.

   After you create an environment, you must invite users to the new environment.

## Delete an environment

## Procedure

1. Log in to your Planning Analytics as a Service instance.
2. On the Planning Analytics as a Service home page, click the **Administration** tile.
3. On the **Administration** page, click the **Environments** tab.
4. Click **Add** and enter the environment name.
5. Click **Delete** for the environment you want to delete.

# Edit an environment name

You can edit an environment name at any time after initial creation.

## About this task

You can edit an environment name only on IBM Planning Analytics as a Service.

## Procedure

1. Click the **Administration** tile on the Planning Analytics Workspace home page.
2. Click the **Environments** tile.
3. Click the **Edit environments** icon for the environment you want to rename.
4. Enter a new environment name, then click **Save**.

## Results

The new environment name is immediately available on the Environments page and on the environments switcher.

# Manage databases on Planning Analytics as a Service

There are some variations in the capabilities and procedures when managing databases on Planning Analytics Administration.

For information on general database administration, see “Monitor and administer databases” on page 914.

## Create a database on Planning Analytics as a Service

You must have the Administrator role to create a database.

### About this task

When you specify the Planning Analytics database name, ensure that you follow rules regarding special characters, character lengths, and other operating system and process restrictions. The database name cannot:

- Have more than 75 characters
- Be blank
- Contain only spaces, and begin or end with a space
- Contain only periods (for example, '...'), and end with a period '.'
- Contain only the number 0
- Contain the following characters: \/":|<>+=;,?*@$ ` '}{%
- Contain the new line (\n) character

### Procedure

1. Open Planning Analytics Workspacein Planning Analytics as a Service.
2. Click the **Administration** tile.
3. Click **Databases**.
4. Click **Add**.
5. Enter a **Database name**.
6. If you want to enable failover protection and ensure maximum database availability, enable the **High availability** option and specify the number of **Replicas** you want to create for the database.

   One (1) or more replicas ensure high availability for the database. It might be desirable to increase the number of replica databases during a critical period of business activity to temporarily improve resilience. You can later reduce the number of replicas during normal operations.

   Each replica counts against your disc space and memory quota.

   Cellsets that are generated by views and MDX execution are balanced across the replicas. This provides a partial load balancing effect among replicas. Cellsets are used by the Planning Analytics clients such as Planning Analytics Workspace and Planning Analytics for Excel to support users changing cell values in the cube viewer and in Excel reports. Metadata changes happen on the leader database node and are propagated to follower replicas in near real time.
7. Specify the amount of disc space you want to allocate to the database. The amount of disc space that is available from your quota is indicated to help you decide how much to allocate.
8. Specify the amount of memory you want to allocate to the database. The amount of memory that is available from your quota is indicated to help you decide how much to allocate.
9. The **Automatic backups** option is enabled by default. Automatic daily backups of your database are included as part of your Planning Analytics as a Service agreement. However, if automatic backups are not desired, as might be the case in a development environment, you can disable the **Automatic backups** option.

10. If backups are enabled, specify a **Backup start time**. This is the time at which backups start each day. The time is specified in local time, but stored in GMT.
11. Click **Create**.

## Upgrade TM1 Database 12 databases

You can upgrade a TM1 Database 12 database directly from Planning Analytics Administration.

### About this task

You can upgrade databases using this procedure only on IBM Cloud Pak for Data and IBM Planning Analytics as a Service.

An upgrade applies to the main (leader) database and any replicas.

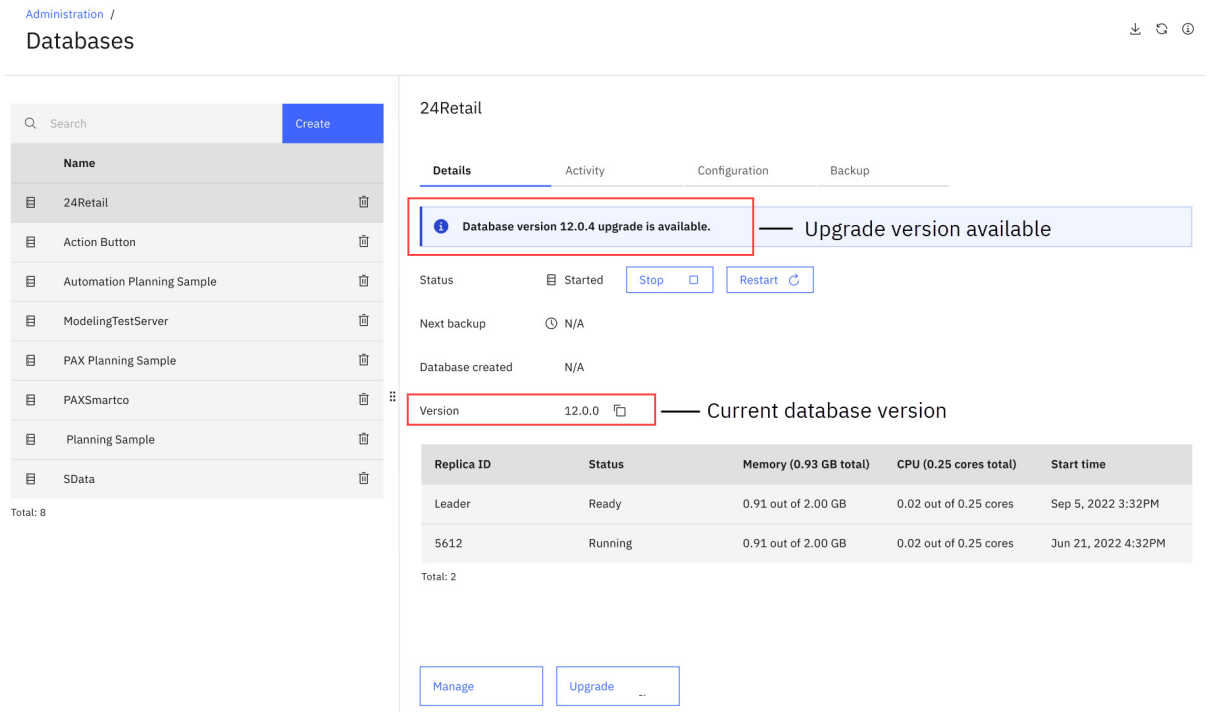
It is not necessary to stop a database prior to upgrading, but the upgrade process runs more efficiently on a stopped database.

**Important:** A database upgrade cannot be undone. You cannot downgrade to a previous version after upgrading a database.

### Procedure

1. On the Planning Analytics Workspace home page, click the **Administration** tile.
2. Click the **Databases** tile.

   The **Details** tab indicates the current installed version of the database and any replicas. If the installed version is prior to the most recent version of the database, a message indicates that a more recent version is available for upgrade and the **Upgrade** button is enabled.




3. Click **Upgrade**.
4. Click **Upgrade** again to confirm your choice.

   A persistent message on the **Databases** page indicates the status of the upgrade.

## Modify database settings on Planning Analytics as a Service

You can manage database settings when a database is started or stopped. You must have the Administrator role to manage a database.

### Procedure

1. Open Planning Analytics Workspacein Planning Analytics as a Service.
2. Click the **Administration** tile.
3. Click **Databases**.
4. Click the database you want to manage.
5. Click the **Details** tab.
6. Click **Manage**.
7. Modify the Replica, Memory, Automatic backups, and Backup start time as described in "Create a database on Planning Analytics as a Service" on page 957.
8. Click **Save**.

## View and manage database backups on Planning Analytics as a Service

You can view all your available database backups. You can also create a manual backup on-demand, upload a backup, or restore a backup.

### Procedure

1. Open Planning Analytics Workspacein Planning Analytics as a Service.
2. Click the **Administration** tile.
3. Click **Databases**.
4. Click the **Backup** tab.

   You can now proceed with any of these database management tasks.

### *Create a manual backup*

### About this task

When you create a database, you can configure an automatic backup schedule that generates a daily backup of your database. You can also create a manual backup at any time.

### Procedure

1. Click **Create** on the **Backup** tab.
2. Enter a **Backup name**.
3. Click **Create**.

   The backup is created as a `tar.gz` archive and appears as a Manual backup type in the list of available items on the **Backup** tab.

### *Upload a backup*

### About this task

You can upload a backup from your local file system or network. The backup must be a valid `.tgz` or `.tar.gz` Planning Analytics database backup archive; no validation is performed on the archive.

### Procedure

1. Click **Upload database backup** .

2. Drag and drop the backup onto the drop zone or click the **...tap to browse** link to browse to the file.

   **Note:** A known issue that occurs only on the Safari browser prevents you from successfully uploading a `.tgz` archive using the **...tap to browse** link. To work around this issue, use drag and drop instead of browse. Alternatively, you can rename the archive with the `.tar.gz` extension before you upload.
3. Click **Upload**.

   The backup you uploaded is available for restoration and is included in the list on the **Backup** tab.

### *Restore a backup*

#### About this task

All of the manual and automatic backups that are available for restoration are listed on the **Backup** tab.

#### Procedure

1. Click the **Action** button ⋮ for the backup that you want to restore to your database.
2. Click **Restore**.

   The dataset contained on the backup is restored to your database.

## Control logging levels

You can specify logging levels for individual loggers in Planning Analytics Administration on TM1 Database 12. This capability is not available on TM1 Database 11.

#### About this task

The available logging levels are:

**Off**
: No logging.

**Debug**
: Detailed, technical messages that are useful when customer support or engineering needs to debug the database.

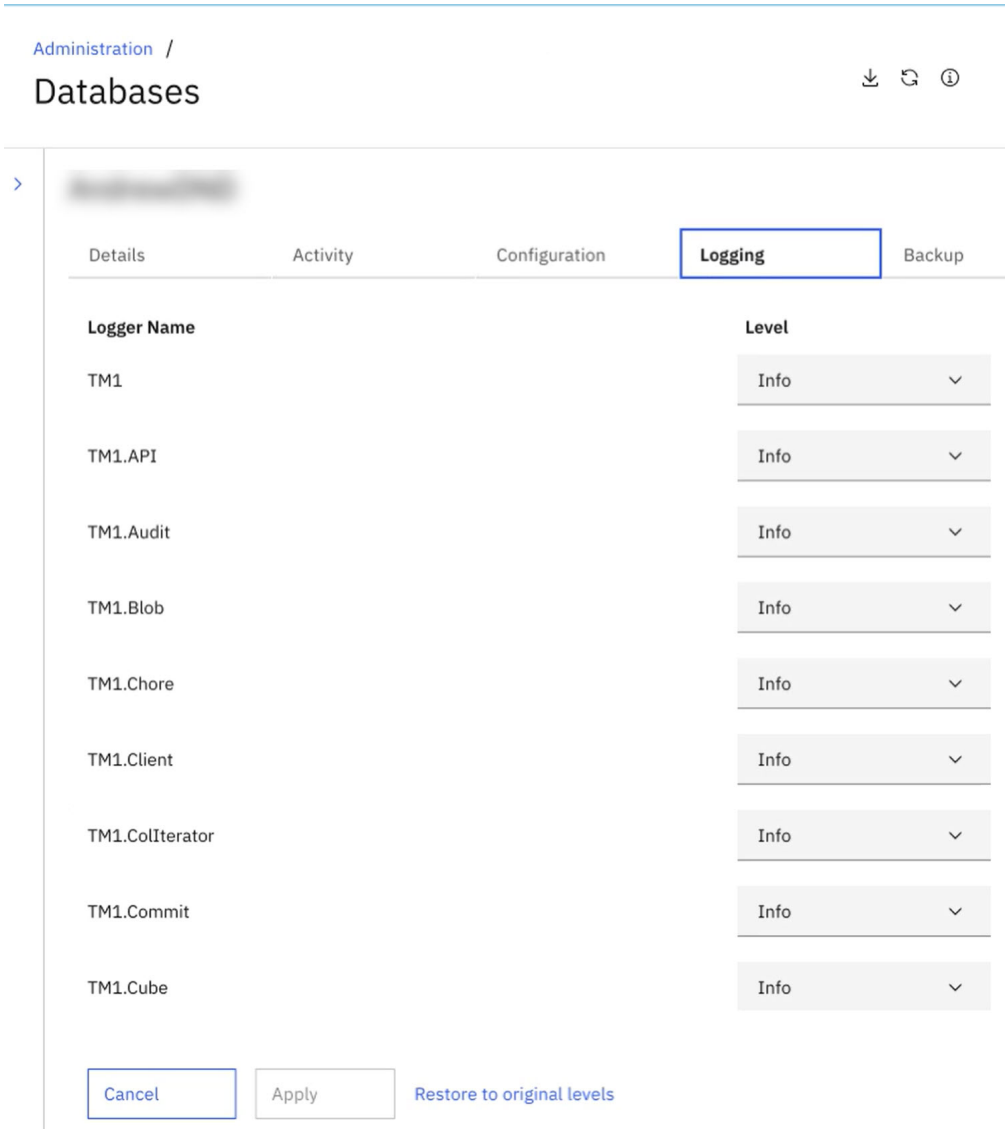
**Info**
: Informational messages that highlight the progress of the database and report normal actions within the database.

**Error**
: An error condition of which you should be aware. Action should be taken to fix or report the issue to customer support.

The default logging level is Info.

#### Procedure

1. Open Planning Analytics Administration.
2. Click the database for which you want to set logging levels.
3. Click the **Logging** tab.




4. To set the level for a logger, click the **Level** menu and select the desired logging level.

   The available loggers are listed alphabetically and hierarchically. If you set a logging level for a parent logger before setting a logging level for any child loggers, all child loggers initially inherit the same level, but can be modified independent of the parent. If you set a logging level for a child prior to setting a logging level for a parent logger, the child logging level remains unchanged when you set the parent level.

   For example, the `TM1` logger is the parent of all other available loggers. Similarly, the `TM1.Event` logger is the parent of the `TM1.Event.Process` and `TM1.Event.Process` loggers.
5. Click **Apply** after you've set all desired logging levels.

## Set permissions for assets

You must have the **Full control** permission on an asset to set permissions for that asset.

The assets include books, views, websheets, applications, plans, workbenches, or folders.

# Set permissions for assets in Planning Analytics Workspace

## About this task

When you set permissions, it is important to understand whether you are modifying permissions that are inherited from the folder where the object resides or whether you are creating a new policy of permissions on this object only.

For example, from a book you can set permissions based on the permissions that are inherited from the folder that the book resides in. This approach means that you use **effective permissions**.

Or you can set permissions specifically for the asset. This approach means that you create a **policy** of explicit permissions.

You can select many assets located in the **Reports and Analysis** and **Data and Models**, and apply permission to them simultaneously.

## Procedure

1. Locate your assets in the **Reports and Analysis**, **Data and Models**, or **Applications and Plans** page.
2. Click the **Shared** tab.
3. Find assets for which you want to set permissions. Use the filter, search, and sort options on the **Shared** tab to narrow the list of items.
4. Click the **Actions** icon ⋮ for assets for which you want to set permissions.
5. Click **Set Permissions**.

   By default, the **Inherit permissions from <parent> content folder** option is selected, indicating that the items inherit permissions from the parent folder. If you leave this option selected, you cannot customize permissions for individual items.
6. Clear the **Inherit permissions from <parent> content folder** option.
7. By default, the **Apply permission changes recursively** option is selected, indicating that permissions applied to the folder, also apply to the content of this folder.
8. By default, you see all users and groups for which you can set permissions on the item, regardless of the current permissions.

   Click the **Filter** icon ▽ to narrow the view by users and groups or by current permission status.
9. For each user or group for which you want to assign permissions, select one of the following options:

   **View only**
   The user or group can view the asset contents and edit data values, but cannot modify asset properties. For example, a user or group with **View only** permission cannot change layout, height, width, position, tabs, or color properties of a book and cannot add, modify, or remove widgets from a book.

   **View and edit**
   The user or group can view the asset contents, edit values, delete the asset, and modify asset properties.

   **Full control**
   The user or group has full administrative control over the asset, and can modify and delete it.

# Set permissions for a book in Planning Analytics Workspace Classic

## About this task

When you set permissions, it is important to understand whether you are modifying permissions that are inherited from the folder where the object resides or whether you are creating a new policy of permissions on this object only.

For example, from a book you can set permissions based on the permissions that are inherited from the folder that the book resides in. This approach means that you use **effective permissions**.

Or you can set permissions specifically for the asset. This approach means that you create a **policy** of explicit permissions.

### Procedure

1. Locate your asset in the **Reports and Analysis** or **Data and Models** page.
2. Click **Set Permissions**.
3. By default, you see **Showing effective permissions**, which means that you are inheriting permissions from the folder where the object is.
   a) Clear **Inherit permissions from folder** and click **Clear all** to reset the permissions and set a policy of permissions for the book.
   b) Click **Inherit permissions from folder** to have the book inherit permissions from the folder that it resides in.

      If you select this option, you can't customize permissions for individual users.
4. Optional: By default, you see all users who you can set permissions for (**All permissions**).
   - To see only users who already have permissions on the asset, select **Assigned only** in the filter beside **All permissions**.
   - To see only users who are not already assigned permissions on the asset, select **Unassigned only** in the filter beside **All permissions**.
5. By default, you see all users and groups that you can set permissions for in **Users and groups**.
   - To see only users to set permissions for, select **Users only** in the filter beside **Users and groups**.
   - To see only groups to set permissions for, select **Groups only** in the filter beside **Users and groups**.
6. For each user that you want to have access to the asset, click the appropriate permission level.

   **View only**
   The user or group can view the asset contents and edit data values, but cannot modify asset properties. For example, a user or group with **View only** permission cannot change layout, height, width, position, tabs, or color properties of a book and cannot add, modify, or remove widgets from a book.

   **Edit only**
   The user or group can view the asset contents, edit values, delete the asset, and modify asset properties.

   **Full control**
   The user or group has full administrative control over the asset, and can modify and delete it.

# Set permissions on the Shared folder and subfolders

If you are an administrator in Planning Analytics Workspace, or a user with full control permissions, you can set user permissions on the Shared folder. Additionally, administrators and other users with required privileges can set user permissions on subfolders within the Shared folder.

## Set permissions on the Shared folder and subfolders in Planning Analytics Workspace

### Before you begin

**Note:** It is recommended that the **Everyone** group is assigned the **View** permission on the root of the shared folder.

## About this task

Because the Shared folder is the highest level of where you can set permissions, this approach is called setting a **policy** for the Shared folder. This **policy** affects all objects in the Shared folder.

## Procedure

1. Click the **Reports and Analysis** tile on the **Home** page.
   a) Click the **Shared folder** tab.
   b) Click the **Set permissions** icon.
   c) On the **Set permissions** dialog box, by default you see all users and groups for which you can set permissions, regardless of the current permissions.

      Click the **Filter** icon to narrow the view by users and groups or by current permission status.
   d) For each user or group for which you want to set permissions, select one of the following **Permission** options:
      - **View only** - The user can view content in the folder, but cannot save content to the folder.
      - **View and edit** - The user can view and add content to the folder, and can also set permissions on content that the user adds to the folder, but cannot set permissions on the folder itself.
      - **Full control** - The user has full control over maintenance of the folder and its contents. For a subfolder anywhere under the Shared folder, the user can move, delete, or rename the subfolder.
   e) Click **Save**.
2. To set permissions for any subfolder within the Shared folder:
   a) Click the **Reports and Analysis** tile on the **Home** page.
   b) Click the **Shared** tab on the **Reports and Analysis** page.
   c) Click the **Filter** icon and select only the **Folder** option.
      You can now see all the folders within the Shared folder.
   d) Click the **Actions** icon for the folder on which you want to set permissions.
   e) Click **Set permissions**.

      By default, the **Inherit permissions from Shared content folder** option is selected, indicating that the folder inherits permissions from the Shared folder. If you leave this option selected, you cannot customize permissions for individual users or groups.
   f) Clear the **Inherit permissions from Shared content folder** option
   g) By default, you see all users and groups for which you can set permissions, regardless of the current permissions.

      Click the **Filter** icon to narrow the view by users and groups or by current permission status.
   h) By default, the **Apply permission changes recursively** option is selected, indicating that permissions applied to the subfolder, also apply to the content of this subfolder.
   i) For each user or group for which you want to set permissions, select one of the following **Permission** options:
      - **View only** - The user can view content in the folder, but cannot save content to the folder.
      - **View and edit** - The user can view and add content to the folder, and can also set permissions on content that the user adds to the folder, but cannot set permissions on the folder itself.
      - **Full control** - The user has full control over maintenance of the folder and its contents. For a subfolder anywhere under the Shared folder, the user can move, delete, or rename the subfolder.
   j) Click **Save**.

# Set permissions on the Shared folder and subfolders in Planning Analytics Workspace Classic

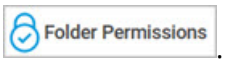
## Before you begin

**Note:** It is recommended that the **Everyone** group is assigned the **View** permission on the root of the shared folder.

## About this task

Because the Shared folder is the highest level of where you can set permissions, this approach is called setting a **policy** for the Shared folder. This **policy** affects all objects in the Shared folder.

## Procedure

1. To set permissions for the Shared folder:
   a) Click the **Shared** folder on the Welcome page, then click 
   
   .

      **Note:** The **Folder Permission** icon is available only when the Shared folder is selected.
   b) On the **Set permissions** dialog box, by default you see all users who you can set permissions for (**All permissions**).
      - To see only users who already have permissions on the Shared folder, select **Assigned only** in the filter beside **All permissions**.
      - To see only users who are not already assigned permissions, select **Unassigned only** in the filter beside **All permissions**.

   c) By default, you see all users and groups that you can set permissions for (**Users and groups**).
      - To see only users to set permissions for, select **Users only** in the filter beside **Users and groups**.
      - To see only groups to set permissions for, select **Groups only** in the filter beside **Users and groups**.

   d) Click the user or group that you want to set permissions for, then click one of the following permissions:
      - **View** - The user can view content in the folder, but cannot save content to the folder.
      - **Edit** - The user can view and add content to the folder, and can also set permissions on content that the user adds to the folder, but cannot set permissions on the folder itself.
      - **Full control** - The user has full control over maintenance of the folder and its contents. For a subfolder anywhere under the Shared folder, the user can move, delete, or rename the subfolder.

   e) Click **OK**.
2. To set permissions for any subfolder within the Shared folder:
   a) Open the parent folder that contains the subfolder for which you want to set permissions.
   b) On the subfolder tile, click ⋮.
   c) Click **Set permissions** and clear the **Inherit permissions from folder 'parent'** option.

      By default, the **Inherit permissions from folder 'parent'** option is selected, indicating that the subfolder inherits permissions from the parent folder. If you leave this option selected, you cannot customize permissions for individual users.
   d) On the **Set permissions** dialog box, by default you see all users who you can set permissions for (**All permissions**).
      - To see only users who already have permissions on the folder, select **Assigned only** in the filter beside **All permissions**.

- To see only users who are not already assigned permissions on the folder, select **Unassigned only** in the filter beside **All permissions**.

e) By default, you see all users and groups that you can set permissions for (**Users and groups**).

- To see only users to set permissions for, select **Users only** in the filter beside **Users and groups**.
- To see only groups to set permissions for, select **Groups only** in the filter beside **Users and groups**.

f) Click the user or group that you want to set permissions for.

g) Click one of the available permissions: **View**, **Edit**, or **Full control**.

h) Click **OK**.

# Limit access to the Set Editor

If you are an administrator, you can enable or disable access to the Set Editor from a view or from a selector that is created from a dimension tile.

By default, users can access the Set Editor from a dimension tile in a view or from a selector that is created from a dimension tile. When you disable access to the Set Editor, users cannot open the Set Editor, but they can select from members of the current set.

## Limit access to the Set Editor Planning Analytics Workspace

### Procedure

1. Click a view or a selector in a book.
2. Click the **Properties** icon .
3. Click the **Custom** tab.
4. Click **Set editor**.
5. Enable or disable the **Allow access to Set editor** option.

# Unload and reload a cube from memory

You can unload a cube from memory to temporarily reduce RAM consumption or to assist in the development and troubleshooting of rules feeders. Unloading a cube also unloads any views of the cube from memory.

### About this task

While a cube is unloaded, any reference to or request for data in the cube will cause the cube to be automatically reloaded, maintaining data availability.

You must be an administrator or modeler to unload or reload cubes.

### Procedure

1. Make sure that you are in edit mode.
2. In the Data tree, right-click the cube you want to unload, then click **Unload cube**.
3. Click **OK** when prompted for confirmation.

   To manually reload a cube into memory, right-click the cube, then click **Reload cube**.

# Access the S:/ drive and files in the remote server

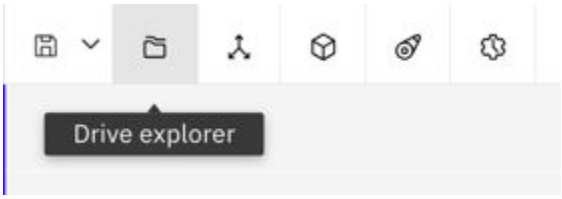
With the **Drive explorer** in the modeling workbench, you can access the S: / drive, and work with files in the root folder. You can also move files to or from the remote server, and copy files to the S: / drive.

**Note:** The **Drive explorer** is not available in IBM Planning Analytics as a Service.

The **Drive explorer** option is visible to users with the subscription administrator access and the Drive explorer capability.

Administrators grant users access to the **Drive explorer** capability. The **Drive explorer** capability uses the same access controls that the Planning Analytics shared folder uses to restrict access to the file system. For more information, see Controlling user access to shared folders.

Click the **Drive explorer** icon in the workbench toolbar to open the root of the S: / drive.




Like in the **File manager**, you can upload files, create folders, and select files in the **Drive explorer**. Selected files can be renamed, moved to a new location, copied, deleted, and compressed.

Use the **Drive explorer** for the following operations:

**Connect to the remote server associated with your Cloud environment**

Once connected, you can move files to a new location.

1. From the **Drive explorer** toolbar, click the **Open remote server** icon.



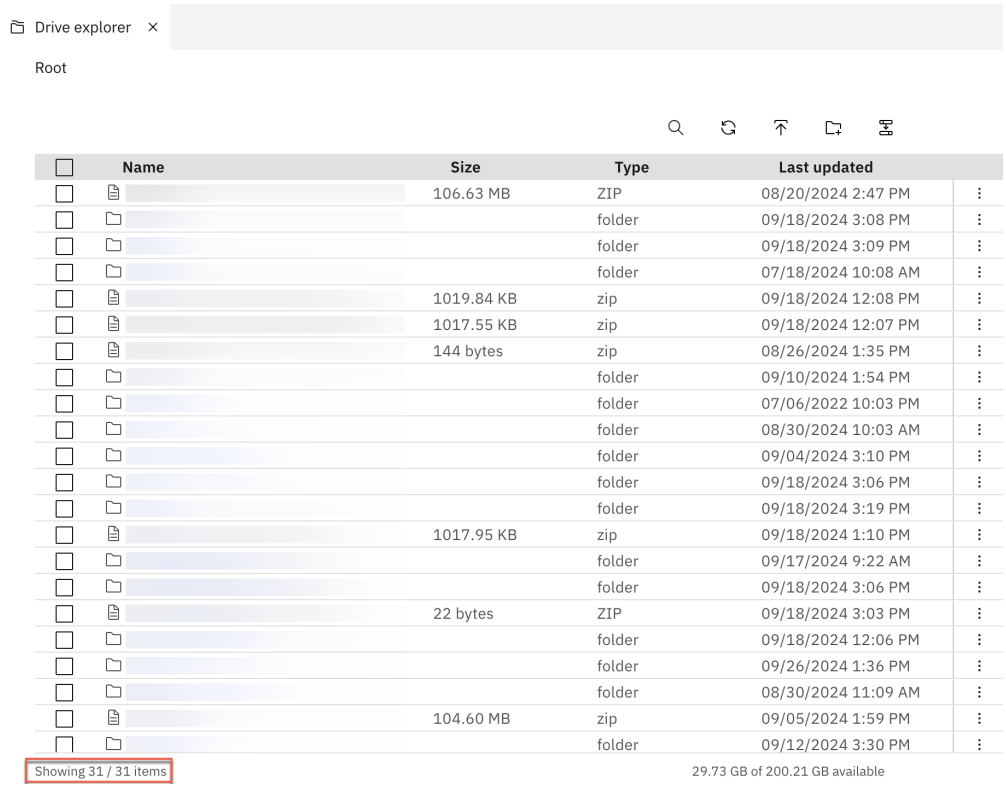

2. Log in to your remote server with your Fileshare credentials, and click **Refresh folder items** to test the connection.

   The credentials to connect to the remote server are provided in the welcome kit for that specific Planning Analytics system. You can now see the files on your remote server.

   You can create folders here to organize files, and select files to rename, delete, and move to a new location. Keep in mind that when you copy a file from the remote server, you can paste the file only to the local S: / drive. You can also copy files from the S: / drive and paste them to the remote server.

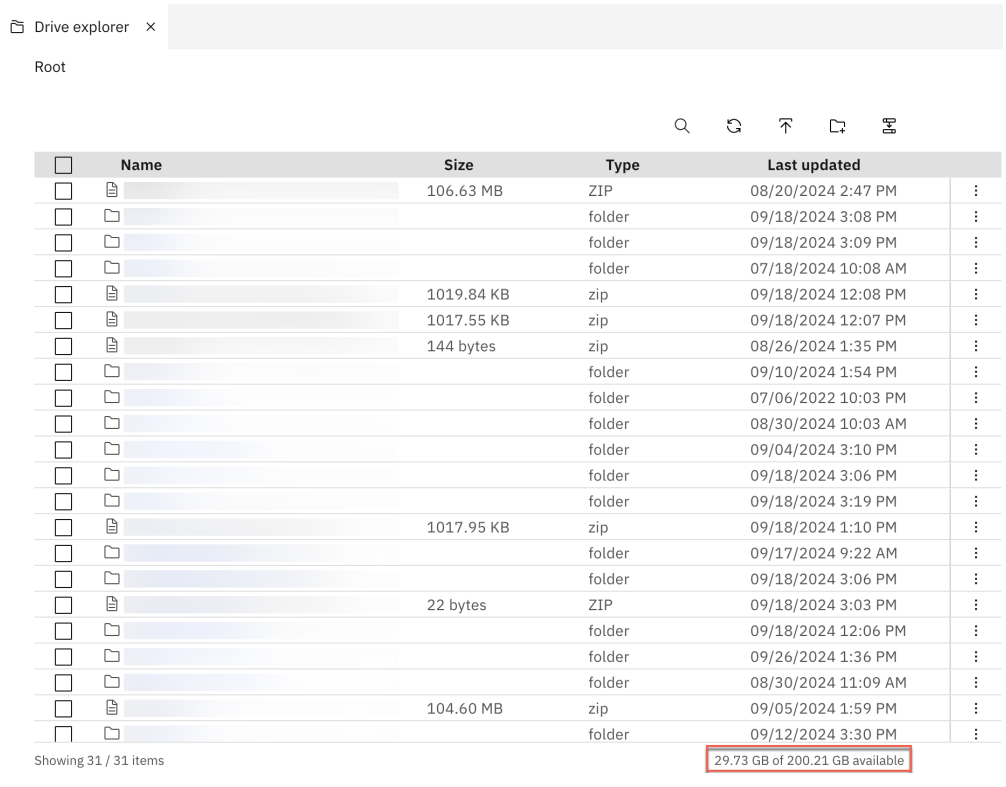
**View the number of files in a directory**

When you are in a directory, you can view the number of items in that directory. This feature is helpful for providing at-a-glance information about the files in large directories. For example, you can see the number of log files that you have in a logging directory.



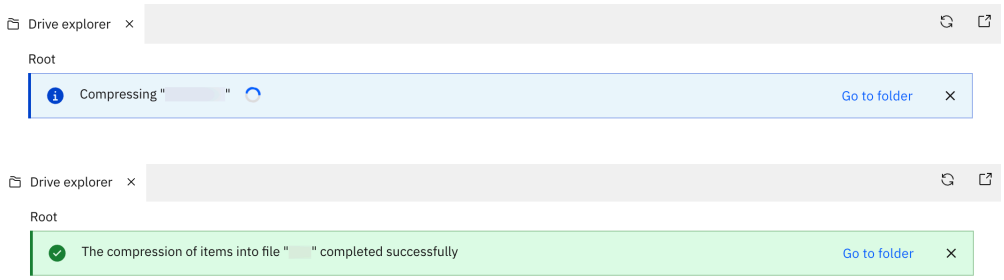

## View disk space

You can view the total amount of disk space that you have, and the amount of disk space that you is left.




## Inline notifications

The drive explorer displays inline notifications about the status of actions that you perform. For example, when compressing files, an inline notification appears when the process is being performed, and when it is completed.




**Unzipping files**

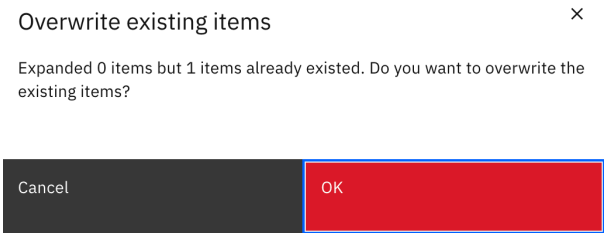
You can expand a file into the directory that you are viewing, or into a folder.

To expand a file into the directory that you are viewing, click **Expand**.

To expand a file into a folder, click **Expand to**, name the folder, and click **Create**. The contents of the unzipped file are in the newly created folder.

**Overwrite notification when expanding a file**

If you expand a file to an existing folder with the same name, the following notification appears:



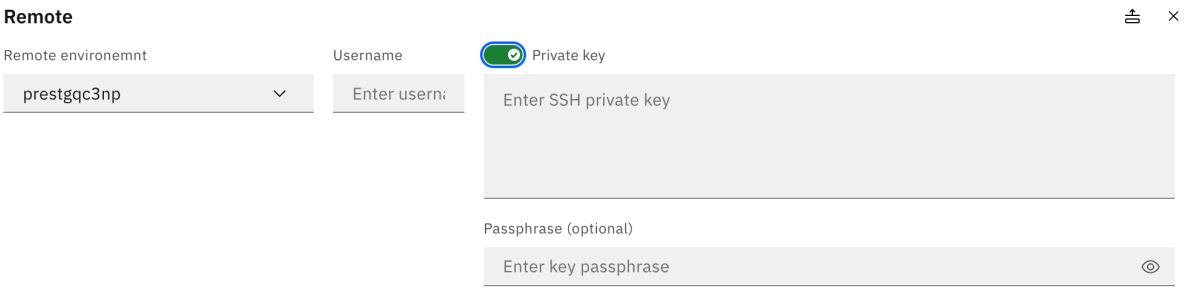

You cannot choose which files are overwritten. The notification appears only once.

**Tm1s.log**

The `tm1s.log` file is visible in the directory. However, the file cannot be opened or deleted.

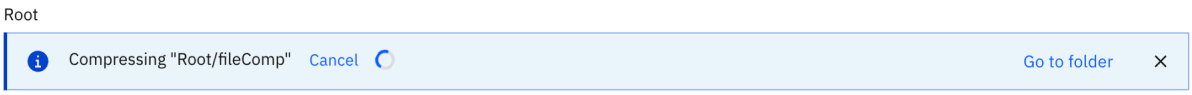
**Support for private keys and passphrases in Secure File Transfer Protocol (SFTP)**

You can use private keys or passphrases in Secure File Transfer Protocol (SFTP). In the **Remote environment** tab in the drive explorer, click the **Private key** toggle. You can then enter a **Private key** or **Passphrase**.



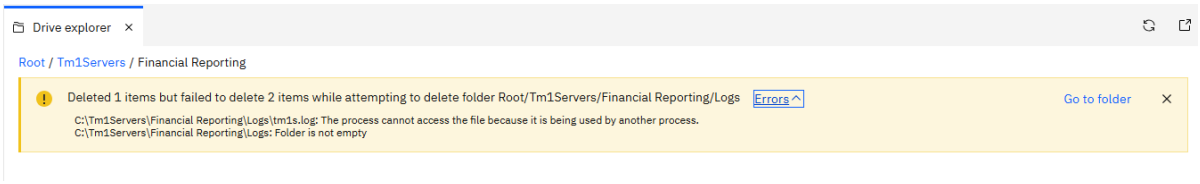

**Cancel current operation**

Using inline notifications, you can cancel an operation while it is being performed. When an inline notification displays the status of an operation, click **Cancel** to stop the process.




**Ignore locked files during copy, move, delete, and compress operations**

When you perform copy, move, delete, and compress operations, locked files do not prevent these operations from completing. Instead, after these operations are completed, a message is displayed which lists the locked files that were not included in the operation.




**Duplicate tab**

You can open a duplicate Drive explorer tab. The duplicate tab opens to the current directory in the original instance of the Drive explorer and reuses your existing credentials. You can use the duplicate tab to browse or manage files in parallel without losing your place, but you cannot copy or paste files between the different Drive explorer tabs.

# Appendix A. References

Use the following resources to learn more about functions and other capabilities that can extend the functionality of Planning Analytics.

## Rules functions

This section contains a complete list of all Planning Analytics rules functions. You can use any of these functions when writing rules.

### Arithmetic operators in Planning Analytics rules

The following mathematical operators can be used when constructing rules.

| Operator | Meaning |
|---|---|
| + (plus sign) | Addition |
| - (minus sign) | Subtraction |
| * (asterisk) | Multiplication |
| / (forward slash) | Division by zero using this operator returns an undefined value. |
| \ (back slash) | Division by zero using this operator returns zero. |
| ^ (caret/circumflex) | Exponentiation |

### Comparison operators in Planning Analytics rules

The comparison operators compare values in the formula portion of a rule calculation statement.

To compare two string values, insert the @ symbol before the comparison operator. For example, `IF ('A' @= 'B',0,1)` yields the number 1.

| Operator | Meaning |
|---|---|
| > | Greater than |
| < | Less than |
| >= | Greater than or equal to |
| <= | Less than or equal to |
| = | Equal to |
| <> | Not equal to |

### Logical operators in Planning Analytics rules

You can combine expressions in a rules calculation statement using logical operators.

© Copyright IBM Corp. 2009, 2025

| Operator | Meaning | Example |
|---|---|---|
| & (ampersand) | AND | (Value1 > 5) & (Value1 < 10) Returns TRUE if the value is greater than 5 and less than 10. |
| % (percentage sign) | OR | (Value1 > 10) % (Value1 < 5) Returns TRUE if the value is greater than 10 or less than 5. |
| ~ (tilde) | NOT | ~(Value1 > 5) Equivalent to (Value1 <= 5) |

# Attribute rules functions

Rules functions that work on attributes.

## ATTRN

ATTRN returns a numeric attribute for a specified element of a dimension or a dimension hierarchy.

This function is valid in both rules and processes.

### Syntax

```
ATTRN(dimension, element, attribute)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element | An element of the dimension or a hierarchy/element pair.<br>If specifying a hierarchy/element pair, you must use this syntax: `'hierarchy_name:element_name'`. Note that the hierarchy name and element name are enclosed in one set of single quotes. |

| Argument | Description |
|---|---|
| attribute | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the element.<br>**Note:** : When this function is used in a conditional statement (IF), the statement is the portion containing the condition, not the entire conditional block. After a minor error, execution continues with the next statement. TurboIntegrator processing has no knowledge that it was in a conditional once the minor error is processed, so the next statement is the next line, not the line after the endif.<br>To avoid this situation, use variables for any operation that could encounter a minor error and then use the variables in the conditional statement. For example:<br>`V1 = CELLGETN('PNLCube', 'fred', 'argentina','Sales','Jan'); IF(V1 = 454);ASCIIOUTPUT ('bug.txt', 'if logic not working properly'); ENDIF;` |

**Example**

In this example, the function returns the numeric value of the Engine Size attribute of the L Series 1.8L Sedan element in the New Offerings hierarchy of the Model dimension.

```
ATTRN('Model', 'New offerings:L Series 1.8L Sedan', 'Engine Size')
```

## ATTRS

ATTRS returns a string attribute for a specified element of a dimension or a dimension hierarchy.

This function is valid in both rules and processes.

### Syntax

```
ATTRS(dimension, element, attribute)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element | An element of the dimension or a hierarchy/element pair.<br>If specifying a hierarchy/element pair, you must use this syntax: `'hierarchy_name:element_name'`. Note that the hierarchy name and element name are enclosed in one set of single quotes. |

| Argument | Description |
|---|---|
| attribute | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the element. |

**Example**

In this example, the function returns the string value of the Currency attribute of the 10100 element in the EMEA hierarchy of the plan_business_unit dimension.

```
ATTRS('plan_business_unit', 'EMEA:10100', 'Currency')
```

## CubeATTRN

CubeATTRN returns a numeric attribute for a specified cube.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
CubeATTRN(CubeName, AttrName);
```

| Argument | Description |
|---|---|
| CubeName | A valid cube name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the cube. |

**Example**

In this example, the function returns the numeric value of the Accounting_Code attribute of the Product cube.

```
CubeATTRN('Product', 'Accounting_Code');
```

## CubeATTRS

CubeATTRS returns a string attribute for a specified cube.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
CubeATTRS(CubeName, AttrName);
```

| Argument | Description |
|---|---|
| CubeName | A valid cube name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the cube. |

### Example

In this example, the function returns the string value of the Owner attribute of the Product cube.

```
CubeATTRS('Product', 'Owner');
```

## DimensionATTRN

DimensionATTRN returns a numeric attribute for a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DimensionATTRN(DimName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

### Example

In this example, the function returns the numeric value of the Accounting_Code attribute of the Plan_Business_Unit dimension.

```
DimensionATTRN('Plan_Business_Unit', 'Accounting_Code');
```

## DimensionATTRS

DimensionATTRS returns a string attribute for a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DimensionATTRS(DimName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

### Example

In this example, the function returns the string value of the Manager attribute of the Plan_Business_Unit dimension.

```
DimensionATTRS('Plan_Business_Unit', 'Manager');
```

## ElementAttrN

ElementAttrN returns a numeric attribute for a specified element of a dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementAttrN(dimension, hierarchy, element, attribute)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | An element of the dimension. |
| attribute | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the element.<br>**Note:** : When this function is used in a conditional statement (IF), the statement is the portion containing the condition, not the entire conditional block. After a minor error, execution continues with the next statement. TI processing has no knowledge that it was in a conditional once the minor error is processed, so the next statement is the next line, not the line after the endif.<br>To avoid this situation, use variables for any operation that could encounter a minor error and then use the variables in the conditional statement. For example:<br>`V1 = CELLGETN('PNLCube', 'fred', 'argentina','Sales','Jan'); IF(V1 = 454);ASCIIOUTPUT ('bug.txt', 'if logic not working properly'); ENDIF;` |

### Example

In this example, the function returns the numeric value of the Engine Size attribute of the L Series 1.8L Sedan element in the Automobile hierarchy of the Model dimension.

```
ElementAttrN('Model', 'Automobile', 'L Series 1.8L Sedan', 'Engine Size')
```

## ElementAttrS

ElementAttrS returns a string attribute for a specified element of a dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementAttrS(dimension, hierarchy, element, attribute)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | An element of the dimension. |
| attribute | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the element. |

### Example

In this example, the function returns the string value of the Currency attribute of the 10100 element in the expense hierarchy of the plan_business_unit dimension.

```
ElementAttrS('plan_business_unit', 'expense', '10100', 'Currency')
```

## Consolidation calculation rules functions

The ConsolidatedMax; ConsolidatedMin; ConsolidatedAvg; ConsolidatedCount; and Consolidated CountUnique perform mathematical calculations on consolidations.

### ConsolidatedAvg

ConsolidatedAvg calculates the average value in a consolidation and returns that single value.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ConsolidatedAvg(flag-value, cube-name, element_1, element_2,… );
```

### Arguments

**flag-value**

The `flag-value` is the **sum** of the following option values:

- 1 - Use weighting when computing the value of consolidated values within the consolidation for which you are determining the average. If this option value is not included in the `flag-value` sum, the raw value of a consolidated element is used.

  The following conditions might affect whether zeros are included in the calculation.

  – If zero is specified as the weighting of some consolidated elements, then the Planning Analytics database configuration parameter `ZeroWeightOptimization=F` must be set for these elements to be included in the calculation of the average value in a consolidation. Without this configuration parameter, the elements for which the weighting is zero are eliminated from the consolidation list, and are therefore not included when calculating the average value in a consolidation.

  – If you want cells containing the value zero to be included when calculating the average, `UNDEFVALS` must be set in the rules for the cube that is specified by the `cube-name` argument. This ensures that when a zero is assigned to a cell of the cube, an actual zero value is stored in the cell and the zero value is included when calculating the average value in a consolidation.

  - If the rules for the cube that is specified by the `cube-name` argument include a SKIPCHECK statement, zeros are *always* ignored when calculating the average value in a consolidation. Remove the SKIPCHECK statement from the rule to include zeros in the consolidation average.
- 2 - Ignore zero values. If this value is included in the `flag-value` sum, zero values will not be included in the calculation of the average value in a consolidation.

There are three valid values for `flag-value`.

- 1 - Use consolidation weighting when computing the consolidation average.
- 2 - Ignore zero values when computing the consolidation average.
- 3 - Use consolidation weighting **and** ignore zero values when computing the consolidation average.

**cube-name**

Name of the cube where the values reside.

If the function is running as part of a cube rule, and NOT as part of a TurboIntegrator process, the cube-name argument can be specified as an empty string to mean the current cube. This means you can write a rule such as `['Apr']=ConsolidatedAvg( 0, '', !actvsbud, '1 Quarter' );`

**element_1, element_2, ...**

Dimension element names that define the intersection of the cube containing the consolidation for which you want to determine the average value.

Arguments *element_1* through *element_n* are sequence-sensitive. *element_1* must be an element from the first dimension of the cube, *element_2* must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables.

#### Example

In a cube that is called Income Statement with three dimensions that are named Regions, Time, and Income Statement, the Income Statement dimension contains an element that is called Gross Sales for the overall sales number.

To calculate the average sales across all regions in the year 2010, write:

```
ConsolidatedAvg( 1,  'Income Statement', 'All Regions', '2010', 'Gross Sales' );
```

## ConsolidateChildren

ConsolidateChildren forces consolidated values to be calculated by summing immediate children along a specified dimension. ConsolidateChildren is useful when intermediate consolidations are calculated by rules and you want a parent consolidation to be calculated by summing the intermediate consolidations rather than by summing the underlying leaf values.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

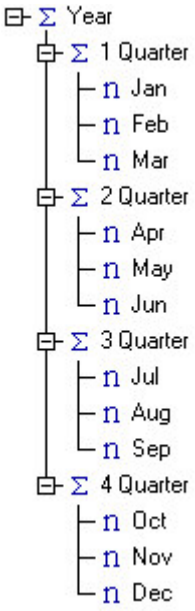
```
ConsolidateChildren(DimName1, DimName2, ...)
```

| Argument | Description |
|---|---|
| DimName1, DimName2, ... | Names of the dimensions along which consolidations will be performed.<br>The function requires at least one DimName argument, and can accept as many DimName arguments as there are dimensions in the cube for which the rule is written. |

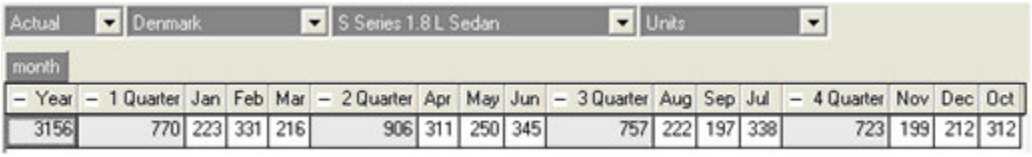
## Example

Consider a cube named Sales composed of the dimensions ActVsBud, Region, Model, Account1, and Month.

In this example, the Month dimension is defined as follows:




If no rule is in place for this cube, the value of the Year consolidation is calculated by summing all the underlying leaf values, in this case Jan through Dec. The following image illustrates this consolidation.

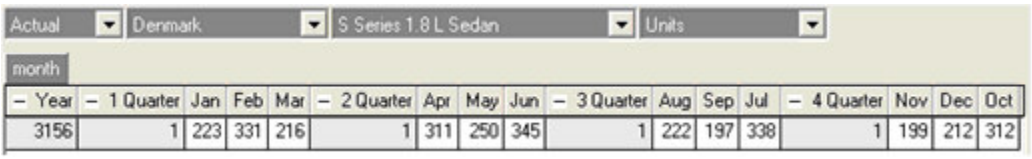



| – Year | – 1 Quarter | Jan | Feb | Mar | – 2 Quarter | Apr | May | Jun | – 3 Quarter | Aug | Sep | Jul | – 4 Quarter | Nov | Dec | Oct |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 3156 | 770 | 223 | 331 | 216 | 906 | 311 | 250 | 345 | 757 | 222 | 197 | 338 | 723 | 199 | 212 | 312 |

Now, suppose you create the following rule for this cube, which indicates that all quarterly values should be 1:

```
[{'1 Quarter', '2 Quarter', '3 Quarter', '4 Quarter'}]=1;
```

The result is as follows:



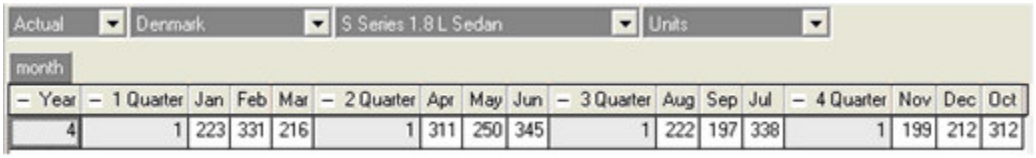

| – Year | – 1 Quarter | Jan | Feb | Mar | – 2 Quarter | Apr | May | Jun | – 3 Quarter | Aug | Sep | Jul | – 4 Quarter | Nov | Dec | Oct |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 3156 | 1 | 223 | 331 | 216 | 1 | 311 | 250 | 345 | 1 | 222 | 197 | 338 | 1 | 199 | 212 | 312 |

In the following image, you can see that quarterly values are indeed calculated by the rule, but the Year consolidation is still calculated by summing all underlying leaf values. If this is not your desired calculation path, you can use the ConsolidateChildren function to force TM1 to calculate the Year consolidation by summing its immediate children, specifically 1 Quarter, 2 Quarter, 3 Quarter, and 4 Quarter.

```
['Year']=ConsolidateChildren('Month');[{'1 Quarter', '2 Quarter', '3 Quarter', '4 Quarter'}]=1;
```

In the rule, the statement `['Year']=ConsolidateChildren('Month')` says that the Year consolidation should be calculated by summing the immediate children of Year in the Month dimension.

The following image shows the result of the `['Year']=ConsolidateChildren('Month')` statement:




| – Year | – 1 Quarter | Jan | Feb | Mar | – 2 Quarter | Apr | May | Jun | – 3 Quarter | Aug | Sep | Jul | – 4 Quarter | Nov | Dec | Oct |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 4 | 1 | 223 | 331 | 216 | 1 | 311 | 250 | 345 | 1 | 222 | 197 | 338 | 1 | 199 | 212 | 312 |

Note that the Year consolidation is now calculated by summing its immediate children.

It's important to remember that for a given consolidation, the ConsolidateChildren function applies only to the *immediate* children of the consolidation.

The ConsolidateChildren function can also be used to specify how consolidations are calculated in multiple dimensions, as in the following example:

| Argument | Description |
|---|---|
| ['World','Year']= ConsolidateChildren('Region','Month') | This statement applies the ConsolidateChildren function to both the World and Year consolidations. In this case, World is calculated by summing all its immediate children in the Region dimension, while Year is calculated by summing its immediate children in the Month dimension. |

## ConsolidatedCount

ConsolidatedCount returns the number of values in a consolidation.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ConsolidatedCount(flag-value, cube-name, element_1, element_2,… );
```

### Arguments

**flag-value**

The `flag-value` is the **sum** of the following option values:

- 1 - Use weighting when computing the value of consolidated values within the consolidation for which you are counting values. If this option value is not included in the `flag-value` sum, the raw value of the consolidated element is used.

  The following conditions might affect whether zeros are included in the calculation.

  – If zero is specified as the weighting of some consolidated elements, then the Planning Analytics database configuration parameter `ZeroWeightOptimization=F` must be set for these elements to be included in the count of values in a consolidation. Without this configuration

parameter, the elements for which the weighting is zero are eliminated from the consolidation list, and are therefore not included when counting the number of values in a consolidation.

  - If you want cells containing the value zero to be included when counting the number of values in a consolidation, UNDEFVALS must be set in the rules for the cube that is specified by the cube-name argument. This ensures that when a zero is assigned to a cell of the cube, an actual zero value is stored in the cell and the zero value is included when counting the number of values in a consolidation.
  - If the rules for the cube that is specified by the cube-name argument include a SKIPCHECK statement, zeros are *always* ignored when counting the number of values in a consolidation. Remove the SKIPCHECK statement from the rule to include zeros when counting the number of values in a consolidation.
- 2 - Ignore zero values. If this value is included in the flag-value sum, zero values will not be includedwhen counting the number of values in a consolidation.

There are three valid values for flag-value.

- 1 - Use consolidation weighting when counting the number of values in a consolidation.
- 2 - Ignore zero values when counting the number of values in a consolidation.
- 3 - Use consolidation weighting **and** ignore zero values when counting the number of values in a consolidation.

**cube-name**

Name of the cube where the values reside.

If the function is running as part of a cube rule, and NOT as part of a TurboIntegrator process, the cube-name argument can be specified as an empty string to mean the current cube. This means you may write a rule such as:`['Apr']=ConsolidatedCount( 1, '', !actvsbud, '1 Quarter' );`

**element_1, element_2, ...**

Dimension element names that define the intersection of the cube containing the consolidation for which you want to count the number of values.

Arguments element_1 through element_n are sequence-sensitive. element_1 must be an element from the first dimension of the cube, element_2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables.

## ConsolidatedCountUnique

ConsolidatedCountUnique counts the number of unique elements for which data points actually exist for the specified consolidation. The unique elements are counted along one dimension of the consolidated cell.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ConsolidatedCountUnique( flag-value, unique-along-dimension-name, cube-name,elem_1,
elem_2, . . . );
```

### Arguments

**flag-value**

The flag-value is the **sum** of the following option values:

- 1 - Use weighting when computing the number of unique elements for which data points actually exist. If this option value is not included in the flag-value sum, the raw values of elements within the consolidation are used.

The following conditions might affect whether zeros are included in the calculation.

- If zero is specified as the weighting of some elements, then the Planning Analytics database configuration parameter ZeroWeightOptimization=F must be set for these elements to be included in the calculation of the number of unique elements. Without this configuration parameter, the elements for which the weighting is zero are eliminated from the consolidation list, and are therefore not included when calculating the number of unique elements.
- If you want cells containing the value zero to be included when calculating the number of unique elements with actual values, UNDEFVALS must be set in the rules for the cube that is specified by the cube-name argument. This ensures that when a zero is assigned to a cell of the cube, an actual zero value is stored in the cell and the zero value is included when calculating the number of unique elements with actual values.
- If the rules for the cube that is specified by the cube-name argument include a SKIPCHECK statement, zeros are *always* ignored when calculating the number of unique elements with actual values. Remove the SKIPCHECK statement from the rule to include zeros in the calculation of the number of unique elements with actual values.

- 2 - Ignore zero values. If this value is included in the flag-value sum, zero values will not be included in the calculation of the number of unique elements with actual values.

There are three valid values for flag-value.

- 1 - Use consolidation weighting when computing the number of unique elements with actual values.
- 2 - Ignore zero values when computing the number of unique elements with actual values.
- 3 - Use consolidation weighting **and** ignore zero values when computing the number of unique elements with actual values.

**unique-along-dimension-name**
The dimension along which unique element entries with real data are to be counted.

**cube-name**

Name of the cube where the values reside.

If the function is running as part of a cube rule, and NOT as part of a TurboIntegrator process, the cube-name argument can be specified as an empty string to mean the current cube.

**element_1, element_2, ...**

Dimension element names that define the intersection of the cube which is the consolidated value to be processed.

Arguments element_1 through element_n are sequence-sensitive. element_1 must be an element from the first dimension of the cube, element_2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables.

### Example

In a cube called Income Statement with three dimensions: Regions, Time, and Income Statement, the Income Statement dimension contains an element called Gross Sales for the overall sales number. To count how many regions had some gross sales in the year 2010 write:

```
ConsolidatedCountUnique( 3, 'Regions', 'Income Statement', 'All Regions', '2010', 'Gross
Sales' );
```

This example uses consolidation weighting **and** ignores zero values when computing the number of unique elements with actual values.

## ConsolidatedMax

ConsolidatedMax calculates the maximum value in a consolidation and returns that single value.

This function is valid in both rules and TurboIntegrator processes.

## Syntax

```
ConsolidatedMax(flag-value, cube-name, element_1, element_2,… );
```

## Arguments

**flag-value**

The `flag-value` is the **sum** of the following option values:

- 1 - Use weighting when computing the value of consolidated values within the consolidation for which you are determining the maximum. If this option value is not included in the `flag-value` sum, the raw value of the consolidated element is used.

  The following conditions might affect whether zeros are included in the calculation.

  – If zero is specified as the weighting of some consolidated elements, then the `Tm1s.cfg` configuration parameter `ZeroWeightOptimization=F` must be set for these elements to be included in the calculation of the maximum value in a consolidation. Without this configuration parameter, the elements for which the weighting is zero are eliminated from the consolidation list, and are therefore not included when calculating the maximum value in a consolidation.

  – If you want cells containing the value zero to be included when calculating the average, `UNDEFVALS` must be set in the rules for the cube that is specified by the `cube-name` argument. This ensures that when a zero is assigned to a cell of the cube, an actual zero value is stored in the cell and the zero value is included when calculating the maximum value in a consolidation.

  – If the rules for the cube that is specified by the `cube-name` argument include a `SKIPCHECK` statement, zeros are *always* ignored when calculating the maximum value in a consolidation. Remove the SKIPCHECK statement from the rule to include zeros in the calculation of the maximum value.

- 2 - Ignore zero values. If this value is included in the `flag-value` sum, zero values will not be included in the calculation of the maximum value in a consolidation.

There are three valid values for `flag-value`.

- 1 - Use consolidation weighting when computing the maximum value in a consolidation.
- 2 - Ignore zero values when computing the maximum value in a consolidation.
- 3 - Use consolidation weighting **and** ignore zero values when computing the maximum value in a consolidation.

**cube-name**

Name of the cube where the values reside.

If the function is running as part of a cube rule, and NOT as part of a TurboIntegrator process, the cube-name argument can be specified as an empty string to mean the current cube. This means you may write a rule such as:`['Apr']=ConsolidatedMax( 1, '', !actvsbud, '1 Quarter' );`

**element_1, element_2, ...**

Dimension element names that define the intersection of the cube containing the consolidation for which you want to determine the maximum value.

Arguments element_1 through element_n are sequence-sensitive. element_1 must be an element from the first dimension of the cube, element_2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables.

### Example

Consider a cube called Income Statement with three dimensions, "Area", "Time", and "Income Statement". The Income Statement dimension contains an element "Gross Sales" for the overall sales number.

To calculate the maximum sales across all regions in the year 2010 use:

```
ConsolidatedMax( 1,  'Income Statement', 'All Regions', '2010', 'Gross Sales' );
```

## ConsolidatedMin

ConsolidatedMin calculates the minimum value in a consolidation and returns that single value.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ConsolidatedMin(flag-value, cube-name, element_1, element_2,… );
```

### Arguments

**flag-value**

The `flag-value` is the **sum** of the following option values:

- 1 - Use weighting when computing the value of consolidated values within the consolidation for which you are determining the minimum. If this option value is not included in the `flag-value` sum, the raw value of the consolidated element is used.

  The following conditions might affect whether zeros are included in the calculation.

  – If zero is specified as the weighting of some consolidated elements, then the Planning Analytics database configuration parameter `ZeroWeightOptimization=F` must be set for these elements to be included in the calculation of the minimum value in a consolidation. Without this configuration parameter, the elements for which the weighting is zero are eliminated from the consolidation list, and are therefore not included when calculating the minimum value in a consolidation.

  – If you want cells containing the value zero to be included when calculating the average, `UNDEFVALS` must be set in the rules for the cube that is specified by the `cube-name` argument. This ensures that when a zero is assigned to a cell of the cube, an actual zero value is stored in the cell and the zero value is included when calculating the minimum value in a consolidation.

  – If the rules for the cube that is specified by the `cube-name` argument include a `SKIPCHECK` statement, zeros are *always* ignored when calculating the minimum value in a consolidation. Remove the SKIPCHECK statement from the rule to include zeros in the calculation of the minimum value.

- 2 - Ignore zero values. If this value is included in the `flag-value` sum, zero values will not be included in the calculation of the minimum value in a consolidation.

There are three valid values for `flag-value`.

- 1 - Use consolidation weighting when computing the minimum value in a consolidation.
- 2 - Ignore zero values when computing the minimum value in a consolidation.
- 3 - Use consolidation weighting **and** ignore zero values when computing the minimum value in a consolidation.

**cube-name**

Name of the cube where the values reside.

If the function is running as part of a cube rule, and NOT as part of a TurboIntegrator process, the cube-name argument can be specified as an empty string to mean the current cube. This means you may write a rule such as:`['Apr']=ConsolidatedMin( 1, '', !actvsbud, '1 Quarter' );`

**element_1, element_2, ...**

Dimension element names that define the intersection of the cube containing the consolidation for which you want to determine the minimum value.

Arguments element_1 through element_n are sequence-sensitive. element_1 must be an element from the first dimension of the cube, element_2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables.

# Cube data rules functions

Rules functions that work on cube data.

## CellValueN

CellValueN returns the numeric value of the specified elements in a cube. This function is valid only in rules. Use of this function in a TurboIntegrator process will result in an error. Use of this function in a TurboIntegrator process will result in an error.

For dimensions not among the element parameters, coordinates are retrieved from the rule target (the cell being retrieved and triggering rule evaluation). The function behavior is analogous to the intra-cube reference expression (e.g. [ 'Measures':'Count' ] ), as used in the formula component of a rule .

The element parameters may be specified in any order, and for CellValueN, multiple elements from the same dimension (but different hierarchies of the dimension) may be specified. Since the elements list is not required to be in cube dimension order, it is necessary to dimension-qualify all element parameters. Element parameters from multi-hierarchy dimensions must also be hierarchy-qualified.

### Syntax

```
CellValueN(cube, element1,..., elementN);
```

| Argument | Description |
|---|---|
| cube | Name of the cube. |
| elementN | Element name that defines the cell. A minimum of one element must be specified. |

### Example

```
CellValueS('ForecastCube', 'Products':'ProductsByChannel':'Channel2', 'Measures':'Count');
```

This example returns the numeric value of the specified cell. The Products dimension has multiple hierarchies while the Measures dimension has one hierarchy.

The intra-cube reference is restricted to literal parameters, while CellValueN is not. This behavior is analogous to the DB() rules function. The element parameters may be specified using string-valued expressions. For example, the previous Products element parameter could be specified as:

```
'Products' : 'ProductsByChannel' : AttrS( … )
```

Unlike DB() and the intra-cube reference expression, CellValueN element parameters must be either dimension-qualified, or dimension and hierarchy qualified.

## CellValueS

CellValueS returns the string value of the specified element(s) in a cube. This function is valid only in rules. Use of this function in a TurboIntegrator process will result in an error.

For dimensions not among the element parameters, coordinates are retrieved from the rule target (the cell being retrieved and triggering rule evaluation). The function behavior is analogous to the intra-cube reference expression (e.g. [ 'Measures':'Count' ] ), as used on formula portion of a rule statement.

The element parameters may be specified in any order, and for CellValueS, multiple elements from the same dimension (but different hierarchies of the dimension) may be specified. Since the elements list is not required to be in cube dimension order, it is necessary to dimension-qualify all element parameters. Element parameters from multi-hierarchy dimensions must also be hierarchy-qualified.

### Syntax

```
CellValueS(cube, element1,..., elementN);
```

| Argument | Description |
|---|---|
| cube | Name of the cube. |
| elementN | Element name that defines the cell. A minimum of one element must be specified. |

### Example

```
CellValueS('ForecastCube', 'Products':'ProductsByChannel':'Channel2', 'Measures':'Location');
```

This example returns the string value of the specified cell. The Products dimension has multiple hierarchies while the Measures dimension has one hierarchy.

The intra-cube reference is restricted to literal parameters, while CellValueS is not. This behavior is analogous to the DB() rules function. The element parameters may be specified using string-valued expressions. For example, the previous Products element parameter could be specified as:

```
'Products' : 'ProductsByChannel' : AttrS( … )
```

Unlike DB() and the intra-cube reference expression, CellValueS element parameters must be either dimension-qualified, or dimension and hierarchy qualified.

## DB

DB returns a value from a cube in a Planning Analytics database. DB returns a numeric value if used in a numeric expression and a string value if used in a string expression.

This function is valid only in rules. Use of this function in a TurboIntegrator process will result in an error.

### Syntax

```
DB(cube, e1, e2, [...e256])
```

### Parameters

**cube**

The name of the cube from which to retrieve the value.

**e1,...en**

Dimension element names that define the intersection containing the value to be retrieved.

Arguments e1 through en are sequence-sensitive. e1 must be an element from the first dimension of the cube, e2 must be an element from the second dimension, and so on.

### Example

In this example, Budget is the cube name, and the function returns the value at the intersection of California, 15" Flat Panel Monitors, Net Sales, and January.

```
DB('Budget', 'California', '15" Flat Panel Monitors', 'Net Sales', 'January')
```

When used to reference multi-hierarchy dimensions, you must specify the particular hierarchy. In this example, the Category2 element exists in the ByCategory hierarchy of the ProductsCube dimension.

```
DB('ProductsCube', 'ByCategory':'Category2',...)
```

### Related information

Using cube references

## ISLEAF

ISLEAF returns 1 if a specified cell is a leaf cell (identified solely by leaf/simple elements). If the specified cell is identified by any consolidated elements, the function returns 0.

The ISLEAF function cannot be used in TurboIntegrator processes. The presence of this function in a process will prevent the process from compiling.

### Syntax

```
ISLEAF
```

### Arguments

None.

### Example

You can use ISLEAF in an IF statement to test if a current cell is a leaf cell. For example:

```
[]=IF((ISLEAF=1),TrueStatement, FalseStatement);
```

Executes the TrueStatement if the current cell is a leaf cell, otherwise it executes the FalseStatement.

## ISUNDEFINEDCELLVALUE

ISUNDEFINEDCELLVALUE compares the passed value to the default numeric cube value, which is influenced by the presence of the UNDEFVALS declaration in that cube's rule. The function returns 1 if the passed value equals the cube's default value, otherwise the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

ISUNDEFINEDCELLVALUE(*TestValue, <Cube>*)

### Arguments

| Argument | Description |
|---|---|
| *TestValue* | The numerical value to compare against the cube's default value. |

| Argument | Description |
|---|---|
| *Cube* | An optional String argument that specifies the cube whose default value should be compared.<br>When ISUNDEFINEDCELLVALUE is used in a rule, the cube is assumed to be the subject cube unless otherwise specified.<br>When used in a process, a cube should be specified.<br>If the cube is omitted in a process, or is not valid when specified, 0 will be used for comparison. |

**Example**

`ISUNDEFINEDCELLVALUE(TestValue)` returns 1 when TestValue is the special undefined value and is used in the rule of a cube with UNDEFVALS declared.

## UNDEF

UNDEF returns the undefined value. This function can be used to prevent data from being stored in a cube based on a logical test.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
UNDEF
```

### Arguments

None.

**Example**

UNDEF returns the undefined value.

## UNDEFINEDCELLVALUE

UNDEFINEDCELLVALUE returns the default numeric cube value, which is influenced by the presence of the UNDEFVALS declaration in that cube's rule.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

`UNDEFINEDCELLVALUE(<Cube>)`

### Arguments

| Argument | Description |
|---|---|
| *Cube* | An optional String argument that specifies the cube whose default value should be returned.<br>When UNDEFINEDCELLVALUE is used in a rule, the cube is assumed to be the subject cube unless otherwise specified.<br>When used in a process, a cube should be specified.<br>If the cube is omitted in a process, or is not valid when specified, 0 will be returned. |

#### Example

UNDEFINEDCELLVALUE returns 0 when used in the rule of a cube without UNDEFVALS declared, or when used in a process.

UNDEFINEDCELLVALUE returns the special undefined value when used in the rule of a cube with UNDEFVALS declared.

`UNDEFINEDCELLVALUE('ExampleCube')` returns the default value of `ExampleCube` or 0 if `ExampleCube` does not exist.

## UNDEFVALS

Putting UNDEFVALS in the rules for a cube changes the default value for the cube from zero to a *special undefined* value. Like other rules functions, UNDEFVALS applies only to the cube associated with the rule in which the function appears.

This function is valid only in rules. Use of this function in a TurboIntegrator process will result in an error.

Use of UNDEFVALS has ramifications regarding how data is stored in the cube and retrieved.

- **Data Storage**

  For a cube without UNDEFVALS in the rules, the default value is zero. If an attempt is made to store a zero in a cell of the cube, that storage request is ignored, as this is a redundant attempt to store the default value, and it would needlessly consume memory space. Similarly, if a cell already contains a value and the value is deleted, nothing is stored in the cell.

  If however the cube has UNDEFVALS defined in the rules, this makes the default value a *special undefined* value. Now when a zero is stored in a cell of a cube, it is actually stored, just like any other non-zero value.

  The *special undefined* value is only a run-time value, returned from requests for cell values. It is never stored in an actual cell in memory, and is never written to disk. Including UNDEFVALS in the rule for a cube has no effect on memory usage or disk storage, except for cells that actually contain zero as a value. When UNDEFVALS is included in the rule for a cube, zero values in that cube will consume memory space and will be written to disk, just like any other data value. If UNDEFVALS is not specified, zero value cells are not stored in memory nor are they written to disk.

- **Data Retrieval**

  For a cube without UNDEFVALS in the rules, the default value is zero. When a cell is retrieved, and there is no value currently stored for that value in the cube, a value of zero (as the default value) is returned. This means that an application cannot tell whether a cell actually exists and contains zero as the cell value, or whether the cell does not exist (as can be the case with sparse data).

  If however the cube has UNDEFVALS defined in the rules, this make the default value a *special undefined* value. In this case, when a non-existent cell is retrieved, the value retrieved will be this *special undefined* value. This can be used to distinguish a cell that does not exist (*special undefined*

returned) from a cell that exists, but whose value is zero (zero returned). Any client written to run against Planning Analytics, which can encounter a cube with UNDEFVALS set, must be prepared to handle a cell value of this *special undefined* rather than a zero. A client can detect whether a value returned from Planning Analytics is this *special undefined* value with the `TM1ValIsUndefined` API function. For details on the `TM1ValIsUndefined` API function, see the *TM1 API* documentation.

**Note:** This *special undefined* value is not the value returned by the UNDEF() TurboIntegrator function. The value returned by UNDEF() is an undefined value used for such things as an attempt to divide by zero, or take the logarithm of an illegal number, etc.

In TurboIntegrator, for normal arithmetic operations (+, -, *, /, \, ^) and normal arithmetic comparisons (<, >, >=, <=, =, <>), the *special undefined* value is treated as a zero. Because of this, the following code does not work:

```
NoCellVal = UndefinedCellValue( 'cube-name' );
If ( vv = NoCellVal );
```

In this comparison, `NoCellVal`, which is the *special undefined* value for an UNDEFVALS cube, is treated as a zero. This means the comparison is really `If ( vv = 0 )`.

In TurboIntegrator you must use the IsUndefinedCellValue to test if a cell value is the *special undefined* value. For example:

```
 vv = CellGetN( 'cube-name', elements-list);
if ( IsUndefinedCellValue( vv, 'cube-name' ) = 1 );
#the cells does not exist
cell_does_not_exist = 1;
else;
#cell exists
cell_does_not_exist = 0;
Endif;
```

### Syntax

```
UNDEFVALS
```

### Arguments

None.

## Date and time rules functions

Rules functions that work with dates and time.

### DATE

DATE returns the date string in *yy-mm-dd* or *yyyy-mm-dd* format for a given serial number.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DATE(SerialNumber, ReturnFourDigitYear)
```

| Argument | Description |
|---|---|
| *SerialNumber* | A date expressed in serial format. |

| Argument | Description |
|---|---|
| *ReturnFourDigitYear* | An optional Boolean argument that determines whether the DATE function returns a string using two- or four-digit notation for the year.<br><br>If ReturnFourDigitYear is true, the function returns date falling within the range of Jan. 1, 1960 and Dec. 31, 9999, using four-digit notation for the year. Serial date 0 corresponds to Jan. 1, 1960 and serial date 2936549 corresponds to Dec. 31, 9999.<br><br>If ReturnFourDigitYear is false, or if this optional argument is omitted from the DATE function, the function returns a date falling within the range Jan. 1, 1960 and Dec. 31, 2059, using two-digit notation for the year. Serial date 0 corresponds to Jan 1, 1960 and serial date 36524 corresponds to Dec. 31, 2059.<br><br>If ReturnFourDigitYear is false or is omitted and you specify a serial date greater than 36524, the serial date used by the function is determined by the formula n - 36525. For example, if you specify a serial date of 36530, then 36530 - 36525 = 5. In this case, DATE uses 5 as the serial date and returns the date Jan. 6, 1960. |

**Example**

DATE(13947) returns `98-03-09`.

DATE(13947, 1) returns `1998-03-09`.

## DATES

DATES returns a date string, in the form 'yy-mm-dd' or 'yyyy-mm-dd', corresponding to a given year, month, and day.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DATES(year, month, day)
```

| Argument | Description |
|---|---|
| year | A year, expressed in either yy or yyyy format. |
| month | A month, expressed in mm format. |
| day | A day, expressed in dd format. |

**Example**

DATES(98, 2, 10) returns '98-02-10'.

DATES(1998, 2, 10) returns '1998-02-10'.

## DAY

DAY returns a numeric value for the day in a given date string.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DAY(DateString)
```

| Argument | Description |
|---|---|
| DateString | A date string in either YY-MM-DD or YYYY-MM-DD format. |

#### Example

DAY('02-05-25') returns 25.

## DAYNO

DAYNO returns the serial date number corresponding to a given date string.

This function is valid in both rules and TurboIntegrator processes.

**Note:** DAYNO can return serial dates for date strings starting at January 1, 1960 (dates string 1960-01-01 or 60-01-01). For dates after December 31, 2059, you use a four digit year in the date string. For example, the date string for January 5, 2061 would be 2061-01-05.

### Syntax

```
DAYNO('DateString')
```

| Argument | Description |
|---|---|
| DateString | A date string in either YY-MM-DD or YYYY-MM-DD format. |

#### Example

DAYNO('98-03-09') returns 13947.

## MONTH

MONTH returns a numeric value for the month in a given date string.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
MONTH(date)
```

| Argument | Description |
|---|---|
| date | A date string in either YY-MM-DD or YYYY-MM-DD format. |

#### Example

MONTH('02-05-25') returns 5.

## NOW

NOW returns the current date/time value in serial number format.

This function is valid in both rules and TurboIntegrator processes.

**Note:** If you are using NOW as a calculated consolidation value, set `RestrictVolatileValuesFromCache` to `True` in the `tm1s.cfg` configuration file to automatically update the consolidation value whenever you refresh or recalculate.

### Syntax

```
NOW
```

### Arguments

None.

#### Example

```
['current_date'] = C: Now()
```

NOW returns the current date/time value in serial number format.

## TIME

TIME returns a string, in HH:MM format, representing the system time on the Planning Analytics database.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
TIME
```

### Arguments

None.

#### Example

Given a system time of 9:33 AM, TIME returns the string '09:33'.

Given a system time of 9:33 PM, TIME returns the string '21:33'.

## TIMST

TIMST returns a formatted date/time string.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
TIMST(datetime, format, ExtendedYears)
```

| Argument | Modifier/<br>Description |
|---|---|
| datetime | A date/time serial number.<br>The integer part of the number specifies the date, and the decimal part specifies the time within the day. Day number 0 corresponds to '60-01-01'. Negative numbers correspond to prior years. Years in the 21st Century, up to 2059, are represented by years 00 through 59. An hour is 1/24th of a day, a minute 1/60th of an hour, and a second 1/60th of a minute. |
| format | A string that formats the result of the function. All the characters in the format argument appear in the result, except for the following characters, which return date/time component values: |
| | \y<br>the last two digits of the year (97, 98, etc.) |
| | \Y<br>the four digits of the year (1997, 1998, etc.) |
| | \m<br>the two digits of the month (01 through 12) |
| | \M<br>the abbreviation of the month (JAN, FEB, etc.) |
| | \d<br>the two digits of the day (01 through 31) |
| | \D<br>the digit of the day (1 through 31) |
| | \h<br>the hour in military time (00 through 23) |
| | \H<br>the standard hour (1 through 12) |
| | \i<br>the minute (00 through 59) |
| | \s<br>the second (00 through 59) |

| Argument | Modifier/<br>Description |
|---|---|
| | \p<br>a.m. or p.m. |
| ExtendedYears | This optional Boolean parameter specifies whether the function returns a date falling within the range 1960 - 2059 or 1960 - 9999.<br>If ExtendedYears is true, the function returns a date falling within the range of Jan. 1, 1960 and Dec. 31, 9999. Serial date 0 corresponds to Jan. 1, 1960 and serial date 2936549 corresponds to Dec. 31, 9999.<br>If ExtendedYears is false, or if this optional argument is omitted from the TIMST function, the function returns a date falling within the range Jan. 1, 1960 and Dec. 31, 2059. Serial date 0 corresponds to Jan 1, 1960 and serial date 36524 corresponds to Dec. 31, 2059.<br>If ExtendedYears is false or is omitted and you specify a serial date greater than 36524, the serial date used by the function is determined by the formula n - 36525. For example, if you specify a serial date of 36530, then 36530 - 36525 = 5. In this case, TIMST uses 5 as the serial date and returns the date Jan. 6, 1960. |

**Example**

TIMST(366.0000, '\M \D, \Y') returns 'JAN 1, 1961'.

TIMST(366.5000, '\H\p \imin\ssec') returns '12p.m. 00min00sec'.

TIMST(366.1000, 'On \M \D, \Y at \H\p \imin\ssec') returns 'On JAN 1, 1961 at 2a.m. 24min00sec'.

TIMST(11111.1100, 'On \M \D, \Y at \H\p \imin\ssec') returns 'On JUN 3,1990 at 2a.m. 38min24sec'.

## TIMVL

TIMVL returns the numeric value of a component (year, month, etc.) of a date/time value.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
TIMVL(datetime, type, ExtendedYears)
```

| Argument | Modifier and<br>Description |
|---|---|
| datetime | A date and time serial number.<br>The integer part of the number specifies the date, and the decimal part specifies the time within the day. Day number 0 corresponds to '60-01-01.' Negative numbers correspond to prior years. Years in the 21st Century, up to 2059, are represented by years 00 through 59. An hour is 1/24th of a day, a minute 1/60th of an hour, and a second 1/60th of a minute. |
| type | A character that specifies the type of component to be extracted. The following are valid type arguments: |
| | Y<br>year value (1997, 1998, etc.) |
| | M<br>month value (1 through 12) |
| | D<br>day value (1 through 31) |
| | H<br>hour value (0 through 23) |
| | I<br>minute value (00 through 59) |
| | S<br>second value (00 through 59) |

| Argument | Modifier and Description |
|---|---|
| ExtendedYears | This optional Boolean parameter specifies whether the function returns a date falling within the range 1960 - 2059 or 1960 - 9999.<br>If ExtendedYears is true, the function returns a date falling within the range of Jan. 1, 1960 and Dec. 31, 9999. Serial date 0 corresponds to Jan. 1, 1960 and serial date 2936549 corresponds to Dec. 31, 9999.<br>If ExtendedYears is false, or if this optional argument is omitted from the TIMVL function, the function returns a date falling within the range Jan. 1, 1960 and Dec. 31, 2059. Serial date 0 corresponds to Jan 1, 1960 and serial date 36524 corresponds to Dec. 31, 2059.<br>If ExtendedYears is false or is omitted and you specify a serial date greater than 36524, the serial date used by the function is determined by the formula n - 36525. For example, if you specify a serial date of 36530, then 36530 - 36525 = 5. In this case, TIMVL uses 5 as the serial date and returns the date Jan. 6, 1960. |

**Example**

TIMVL(11111.1100, 'Y') returns 1990.

TIMVL(11111.1100, 'H') returns 2.

## TODAY

TODAY returns the current date in yy-mm-dd format.

This function is valid in both rules and TurboIntegrator processes.

## Syntax

```
TODAY(<ReturnFourDigitYear>)
```

| Argument | Description |
|---|---|
| ReturnFourDigitYear | An optional Boolean argument that determines whether the TODAY function returns a string using two- or four-digit notation for the year.<br><br>If ReturnFourDigitYear is true, the function returns date falling within the range of Jan. 1, 1960 and Dec. 31, 9999, using four-digit notation for the year. Serial date 0 corresponds to Jan. 1, 1960 and serial date 2936549 corresponds to Dec. 31, 9999.<br><br>If ReturnFourDigitYear is false, or if this optional argument is omitted from the TODAY function, the function returns a date falling within the range Jan. 1, 1960 and Dec. 31, 2059, using two-digit notation for the year. Serial date 0 corresponds to Jan 1, 1960 and serial date 36524 corresponds to Dec. 31, 2059.<br><br>If ReturnFourDigitYear is false or is omitted and you specify a serial date greater than 36524, the serial date used by the function is determined by the formula n - 36525. For example, if you specify a serial date of 36530, then 36530 - 36525 = 5. In this case, TODAY uses 5 as the serial date and returns the date Jan. 6, 1960. |

**Example**

P1=TODAY(1) returns a data string in YYYY-MM-DD format such as 2009-06-05.

P1=TODAY(0) returns a date string in YY-MM-DD format such as 09-06-05

### YEAR

YEAR returns a numeric value for the year in a given date string.

This function is valid in both Planning Analytics rules and processes.

### Syntax

```
YEAR(date)
```

| Argument | Description |
|---|---|
| date | A date string formatted as either YY-MM-DD or YYYY-MM-DD. |

**Example**

YEAR('02-05-25') returns 2.

## Dimension Information Rules Functions

Rules functions that manage dimension information.

## DIMIX

DIMIX returns the index number of an element within a dimension.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
DIMIX(server_name:dimension, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name qualified by the database name. |
| element | The name of an element within the dimension.<br>If the element is not a member of the dimension specified, the function returns 0. |

#### Example

Brazil has an index value of three in the Region dimension. The example returns 3.

```
DIMIX('planning_sample:Region','Brazil')
```

## DIMNM

DIMNM returns the element of a dimension that corresponds to the index argument.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DIMNM(server_name:dimension, index)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name qualified by the database name. |
| index | A value less than or equal to the number of elements in the dimension.<br>If this argument is less than 1, or greater than the number of elements in the dimension, the function returns 0. |

#### Example

This example returns 'Belgium', which is the element within the Region dimension with an index value of 2.

```
DIMNM(planning_sample:'Region',2)
```

## DIMSIZ

DIMSIZ returns the number of elements within a specified dimension.

This function is valid in rules and TurboIntegrator processes.

### Syntax

```
DIMSIZ(dimension)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name.<br>Some installations may need to qualify the dimension name with the database name, as in `database_name:dimension`. |

#### Example

If the dimension Accounts contains 19 elements, the example returns the value 19.

```
DIMSIZ('Accounts')
```

## DNEXT

DNEXT returns the element name that follows the element specified as an argument to the function.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DNEXT(dimension, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name.<br>Some installations may need to qualify the dimension name with the database name, as in `database_name:dimension`. |
| element | The name of an element within the dimension. This argument can also be the name of an alias for a dimension element. |

#### Example

If the Location dimension contains the ordered elements California, Oregon, and Washington, the example returns Washington.

```
DNEXT("Location","Oregon")
```

## DNLEV

DNLEV returns the number levels in a dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DNLEV(dimension)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name.<br>Some installations may need to qualify the dimension name with the database name, as in `database_name:dimension`. |

#### Example

```
DNLEV('Region')
```

In the Region dimension, the various nations (Level 0) add up to regions (Level 1). The regions then add up to super-regions (Level 2), which in turn add up to the world (Level 3).

There are four levels in the Region dimension, so the example returns the value 4.

## DTYPE

DTYPE returns information about the element type of a specified element. DTYPE returns N if the element is a numeric element, S if the element is a string element, C if the element is a consolidated element.

In the case of an element attribute dimension, DTYPE returns AN if the attribute is a numeric attribute, AS if the attribute is a string attribute, and AA if the attribute is an alias attribute. For more information, see Element Attributes.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
DTYPE(dimension, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element | The name of an element within the dimension. |

#### Example

The element Europe is a consolidated element of the Region dimension, so the example returns C.

```
DTYPE('Region','Europe')
```

## TABDIM

TABDIM returns the dimension name that corresponds to the index argument.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
TABDIM(cube, index)
```

| Argument | Description |
|---|---|
| cube | A valid cube name. |
| index | A positive value less than or equal to the total number of dimensions in the cube. |

**Example**

The cube SalesCube contains five dimensions: account1, actvsbud, model, month, and region. The example returns model, the third dimension of SalesCube.

```
TABDIM('SalesCube',3)
```

# Element Information Rules Functions

Rules functions that manage element information.

## ELCOMP

ELCOMP returns the name of a child of a consolidated element in a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ELCOMP(dimension, element, position)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element | The name of a consolidated element within the dimension. |
| position | A positive value less than or equal to the total number of children in the specified element. |

**Example**

In the dimension Region, the consolidated element Central Europe is a consolidation of the children France and Germany. Germany is in the second position in this consolidation. Accordingly, the example returns Germany.

```
ELCOMP('Region','Central Europe',2)
```

## ELCOMPN

ELCOMPN returns the number of components in a specified element. If the element argument is not a consolidated element, the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ELCOMPN(dimension, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element | The name of a consolidated element within the dimension. |

**Example**

In the Region dimension, the element Scandinavia is a consolidation of three elements. The example returns 3.

```
ELCOMPN('Region','Scandinavia')
```

## ElementComponent

ElementComponent returns the name of a child of a consolidated element in a specified dimension. If the element argument is not a consolidated element, the function returns an empty string.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementComponent(dimension, hierarchy, element, position)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | The name of a consolidated element within the dimension. |
| position | A positive value less than or equal to the total number of children in the specified element. |

**Example**

In the dimension Region, the consolidated element Central Europe is a consolidation of the children France and Germany. Germany is in the second position in this consolidation. Accordingly, the example returns Germany.

```
ElementComponent('Region', 'Europe', 'Central Europe', 2)
```

## ElementComponentCount

ElementComponentCount returns the number of components in a specified element. If the element argument is not a consolidated element, the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementComponentCount(dimension, hierarchy, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | The name of a consolidated element within the dimension. |

**Example**

In the Region dimension, the element Scandinavia is a consolidation of three elements. The example returns 3.

```
ElementComponentCount('Region', '', 'Scandinavia')
```

## ElementCount

ElementCount returns the number of elements within a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementCount(dimension, hierarchy)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name.<br>Some installations may need to qualify the dimension name with the database name, as in `database_name:dimension`. |
| hierarchy | The name of the hierarchy within the dimension. |

**Example**

If the Receivables hierarchy in the Accounts dimension contains 19 elements, the example returns the value 19.

```
ElementCount('Accounts', 'Receivables')
```

## ElementFirst

ElementFirst returns the first element of a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementFirst(database_name:dimension, hierarchy)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |

| Argument | Description |
|---|---|
| hierarchy | The name of the hierarchy within the dimension. |

**Example**

If the North America hierarchy of the Location dimension contains the ordered elements California, Oregon, and Washington, the example returns California.

```
ElementFirst("planning_sample:Location", "North America")
```

## ElementIndex

ElementIndex returns the index number of an element within a dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementIndex(dimension, hierarchy, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | The name of an element within the dimension.<br>If the element is not a member of the dimension specified, the function returns 0. |

### Example

Brazil has an index value of three in the Region dimension. The example returns 3.

```
ElementIndex('Region', 'South America', 'Brazil')
```

## ElementIsAncestor

ElementIsAncestor determines whether element1 is an ancestor of element2 in the specified dimension. The function returns 1 if element1 is an ancestor of element2, otherwise the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementIsAncestor(dimension, hierarchy, element1, element2)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element1 | The name of an element within the dimension. |

| Argument | Description |
|---|---|
| element2 | The name of an element within the dimension. |

**Example**

In the Western hierarchy of the Region dimension, the element Europe is an ancestor of Germany. The example returns 1.

```
ElementIsAncestor('Region', 'Western', 'Europe', 'Germany')
```

## ElementIsComponent

ElementIsComponent determines whether element1 is a child of element2 in the specified dimension. The function returns 1 if element1 is a child of element2, otherwise the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementIsComponent(dimension, hierarchy, element1, element2)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element1 | The name of an element within the dimension. |
| element2 | The name of an element within the dimension. |

**Example**

In the dimension Region, the element Central Europe is a consolidation of two elements, Germany and France. The example returns 1.

**Note:** this function returns 1 only for immediate children. In the previous example, Germany is a child of Central Europe. Further, Central Europe is a child of Europe.

```
ElementIsComponent('Region', 'Countries', 'Germany', 'Central Europe')
```

However, because the function returns 1 only for immediate children, the following example returns 0:

```
ElementIsComponent('Region', 'Countries'', 'Germany', 'Europe')
```

## ElementIsParent

ElementIsParent determines whether element1 is a parent of element2 in the specified dimension. The function returns 1 if element1 is a parent of element2, otherwise the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementIsParent(dimension, hierarchy, element1, element2)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element1 | The name of an element within the dimension. |
| element2 | The name of an element within the dimension. |

**Example**

In the dimension Region, the consolidated element Central Europe is the parent of both Germany and France. Accordingly, the example returns 1.

**Note:** this function returns 1 only for immediate parents. In the previous example, Europe is a parent of Central Europe. Further, Central Europe is a parent of Germany.

```
ElementIsParent('Region', 'Countries', 'Central Europe', 'Germany')
```

However, because Europe is not an immediate parent of Germany, the following example returns 0:

```
ElementIsParent('Region', 'Countries', 'Europe', 'Germany')
```

## ElementLevel

ElementLevel returns the level of an element within a dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

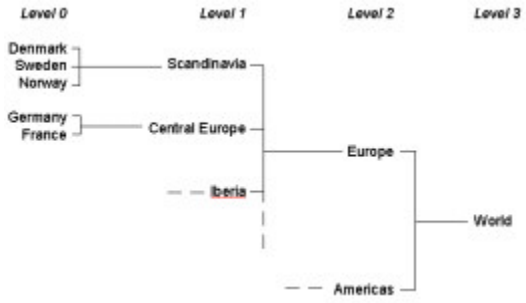
```
ElementLevel(dimension, hierarchy, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | The name of an element within the dimension. |

**Example**

```
ElementLevel('Region','Countries', 'Europe')
```

In the Region dimension, individual nations (Level 0) add up to regions (Level 1). The regions then add up to super-regions (Level 2), which in turn add up to the world (Level 3). The example returns 2, as Europe is a Level 2 element.




## ElementName

ElementName returns the element of a dimension that corresponds to the index argument.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementName(dimension, hierarchy, index)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| index | A value less than or equal to the number of elements in the dimension.<br>If this argument is less than 1, or greater than the number of elements in the dimension, the function returns 0. |

#### Example

This example returns 'Belgium', which is the element within the Countries hierarchy of the Region dimension with an index value of 2.

```
ElementName('Region', 'Countries', 2)
```

## ElementNext

ElementNext returns the element name that follows the element specified as an argument to the function.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementNext(dimension, hierarchy, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name.<br>Some installations may need to qualify the dimension name with the database name, as in `database_name:dimension`. |

| Argument | Description |
|---|---|
| hierarchy | The name of the hierarchy within the dimension. |
| element | The name of an element within the dimension. This argument can also be the name of an alias for a dimension element. |

**Example**

If the Location dimension contains the ordered elements California, Oregon, and Washington, the example returns Washington.

```
ElementNext("Location","Cities", "Oregon")
```

## ElementParent

ElementParent returns the parent of an element in a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementParent(dimension, hierarchy, element, index)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | The name of an element within the dimension. |
| index | A positive value less than or equal to the total number of consolidated elements (parents) that use the element argument as a child. |

**Example**

In the dimension Model, the element Wagon 4WD is a child of both Total Wagons and Total 4WD. Therefore, both Total Wagons and Total 4WD are parents of Wagon 4WD. In the structure of the Model dimension, Total Wagons is defined first, Total 4WD is defined second.

```
ElementParent('Model', 'Automobile', 'Wagon 4WD', 2)
```

The example returns Total 4WD, as this is the second instance of a parent to Wagon 4WD within the Model dimension.

## ElementParentCount

ElementParentCount returns the number of parents of an element in a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementParentCount(dimension, hierarchy, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | The name of an element within the dimension. |

#### Example

In the Model dimension, the element Wagon 4WD is a child of both Total Wagons and Total 4WD. Therefore, both Total Wagons and Total 4WD are parents of Wagon 4WD. The function returns 2.

```
ElementParentCount('Model', 'Automobile', 'Wagon 4WD')
```

## ElementType

ElementType returns information about the element type of a specified element.

ElementType returns N if the element is a numeric element, S if the element is a string element, and C if the element is a consolidated element.

In the case of an element attribute dimension, ElementType returns AN if the attribute is a numeric attribute, AS if the attribute is a string attribute, and AA if the attribute is an alias attribute. For more information, see Element Attributes.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementType(dimension, hierarchy, element)
```

*Table 15. ElementType syntax*

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element | The name of an element within the dimension. |

#### Example

The element Europe is a consolidated element of the Region dimension, so the example returns C.

```
ElementType('Region', 'Countries', 'Europe')
```

## ElementWeight

ElementWeight returns the weight of a child in a consolidated element.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ElementWeight(dimension, hierarchy, element1, element2)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| hierarchy | The name of the hierarchy within the dimension. |
| element1 | The name of a consolidated element within the dimension. |
| element2 | The name of a child of the consolidated element. |

#### Example

The element Variable Costs, which is a child of Gross margin, has a weight of -1. The following example returns -1.

```
ElementWeight('Account1', 'SubAccount1', 'Gross margin', 'Variable Costs')
```

## ELISANC

ELISANC determines whether element1 is an ancestor of element2 in the specified dimension. The function returns 1 if element1 is an ancestor of element2, otherwise the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ELISANC(dimension, element1, element2)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element1 | The name of an element within the dimension. |
| element2 | The name of an element within the dimension. |

#### Example

In the dimension Region, the element Europe is an ancestor of Germany. The example returns 1.

```
ELISANC('Region', 'Europe', 'Germany')
```

## ELISCOMP

ELISCOMP determines whether element1 is a child of element2 in the specified dimension. The function returns 1 if element1 is a child of element2, otherwise the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ELISCOMP(dimension, element1, element2)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element1 | The name of an element within the dimension. |
| element2 | The name of an element within the dimension. |

#### Example

In the dimension Region, the element Central Europe is a consolidation of two elements, Germany and France. The following example returns 1.

**Note:** this function returns 1 only for immediate children. In this example, Germany is a child of Central Europe. Further, Central Europe is a child of Europe.

```
ELISCOMP('Region','Germany','Central Europe')
```

However, because the function returns 1 only for immediate children, the following example returns 0:

```
ELISCOMP('Region','Germany','Europe')
```

## ELISPAR

ELISPAR determines whether element1 is a parent of element2 in the specified dimension. The function returns 1 if element1 is a parent of element2, otherwise the function returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ELISPAR(dimension, element1, element2)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element1 | The name of an element within the dimension. |
| element2 | The name of an element within the dimension. |

#### Example

In the dimension Region, the consolidated element Central Europe is the parent of both Germany and France. Accordingly, the following example returns 1.

**Note:** this function returns 1 only for immediate parents. In this example, Europe is a parent of Central Europe. Further, Central Europe is a parent of Germany.

```
ELISPAR('Region','Central Europe','Germany')
```

However, because Europe is not an immediate parent of Germany, the following example returns 0:

```
ELISPAR('Region','Europe','Germany')
```

## ELLEV

ELLEV returns the level of an element within a dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

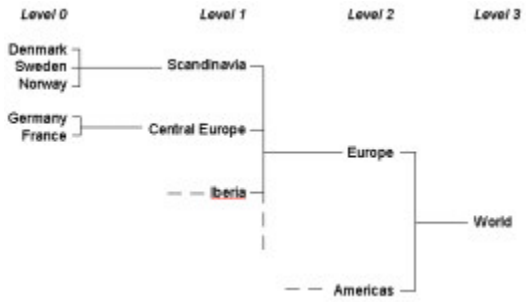
```
ELLEV(dimension, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element | The name of an element within the dimension. |

#### Example

```
ELLEV('Region','Europe')
```

In the Region dimension, individual nations (Level 0) add up to regions (Level 1). The regions then add up to super-regions (Level 2), which in turn add up to the world (Level 3). The example returns 2, as Europe is a Level 2 element.




## ELPAR

ELPAR returns the parent of an element in a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ELPAR(dimension, element, index)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element | The name of an element within the dimension. |
| index | A positive value less than or equal to the total number of consolidated elements (parents) that use the element argument as a child. |

**Example**

In the dimension Model, the element Wagon 4WD is a child of both Total Wagons and Total 4WD. Therefore, both Total Wagons and Total 4WD are parents of Wagon 4WD. In the structure of the Model dimension, Total Wagons is defined first, Total 4WD is defined second.

```
ELPAR('Model','Wagon 4WD',2)
```

The example returns Total 4WD, as this is the second instance of a parent to Wagon 4WD within the Model dimension.

## ELPARN

ELPARN returns the number of parents of an element in a specified dimension.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ELPARN(dimension, element)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element | The name of an element within the dimension. |

**Example**

In the Model dimension, the element Wagon 4WD is a child of both Total Wagons and Total 4WD. Therefore, both Total Wagons and Total 4WD are parents of Wagon 4WD. The function returns 2.

```
ELPARN('Model','Wagon 4WD')
```

## ELWEIGHT

ELWEIGHT returns the weight of a child in a consolidated element.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ELWEIGHT(dimension, element1, element2)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name. |
| element1 | The name of a consolidated element within the dimension. |
| element2 | The name of a child of the consolidated element. |

**Example**

The element Variable Costs, which is a child of Gross margin, has a weight of -1.

The following example returns -1.

```
ELWEIGHT('Account1','Gross margin','Variable Costs')
```

### LevelCount

LevelCount returns the number levels in a dimension.

This function is valid in both rules and TurboIntegrator processes.

#### Syntax

```
LevelCount(dimension, hierarchy)
```

| Argument | Description |
|---|---|
| dimension | A valid dimension name.<br>Some installations may need to qualify the dimension name with the database name, as in `database_name:dimension`. |
| hierarchy | The name of the hierarchy within the dimension. |

**Example**

```
LevelCount('Region', 'Countries')
```

In the Region dimension, the various nations (Level 0) add up to regions (Level 1). The regions then add up to super-regions (Level 2), which in turn add up to the world (Level 3).

There are four levels in the Region dimension, so the example returns the value 4.

## Financial Rules Functions

Rules functions used to manage financial information.

### FV

FV returns the value of an annuity at the time of the last payment. An annuity is a series of payments made at equal intervals of time. Payments are assumed to be made at the end of each period.

This function is valid in both rules and TurboIntegrator processes.

#### Syntax

```
FV(payment, interest, periods)
```

| Argument | Description |
|---|---|
| payment | The amount of the payment made per period. |
| interest | The interest rate paid per period. |
| periods | The number of periods in the annuity. |

### Example

This example returns the value of an annuity at the end of 5 years, with payments of $1,000 per year at 14% interest.

```
FV(1000, .14, 5)
```

## PAYMT

PAYMT returns the payment amount of an annuity based on a given initial value or principal, an interest rate, and a number of periods. An annuity is a series of payments made at equal intervals of time.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
PAYMT(principal, interest, periods)
```

| Argument | Description |
|---|---|
| principal | The present value, or the total amount that a series of future payments is worth now. |
| interest | The interest rate paid per period. |
| periods | The number of periods in the annuity. Payments are assumed to be made at the end of each period. |

### Example

This example returns the payment on a 5-year annuity that is paid yearly, with a principal of $100,000 at 14% interest.

```
PAYMT(100000, .14, 5)
```

## PV

PV returns the initial or principal value of an annuity.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
PV(payment, interest, periods)
```

| Argument | Description |
|---|---|
| payment | The amount of the payment made. |
| interest | The interest rate paid per period. |
| periods | The number of periods in the annuity. Payments are assumed to be made at the end of each period. |

### Example

This example returns the principal value of an annuity with 5 yearly payments of $1,000 at 14% interest.

```
PV(1000, .14, 5)
```

# Hierarchy Rules Functions

Functions to manage hierarchies in rules.

## Hierarchy

If there is only one hierarchy included in the supplied dimension, Hierarchy returns the name of the hierarchy. Otherwise, it returns an empty string. Hierarchy is valid in rules only.

With the introduction of support for multiple hierarchies, it is necessary to identify which hierarchies are in context when multiple hierarchies are being used.

The Hierarchy function cannot be used in TurboIntegrator processes. The presence of this function in a process will prevent the process from compiling.

### Syntax

```
Hierarchy (DimName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |

### Example

This example returns 'Quarter', which is the only hierarchy in the Quarter dimension.

```
Hierarchy ('Quarter');
```

## HierarchyCount

HierarchyCount returns the number of hierarchies in the supplied dimension. HierarchyCount is valid in rules only.

The HierarchyCount function cannot be used in TurboIntegrator processes. The presence of this function in a process will prevent the process from compiling.

### Syntax

```
HierarchyCount (DimName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |

### Example

This example returns 3, which is the number of hierarchies in the model dimension.

```
HierarchyCount ('model');
```

## HierarchyIndex

HierarchyIndex returns a 1-based index if the hierarchy is in the supplied dimension, 0 otherwise. HierarchyIndex is valid in rules only.

HierarchyIndex cannot be used in TurboIntegrator processes. The presence of this function in a process will prevent the process from compiling.

### Syntax

```
HierarchyIndex (DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| HierName | A valid hierarchy name that you want to find the index position of in *DimName*. |

#### Example

This example returns 3, which is the index position of the CustomerTarget hierarchy in the model dimension.

```
HierarchyIndex ('model', 'CustomerTarget');
```

## HierarchyN

HierarchyN returns the name of the hierarchy at a specified position in the supplied dimension and an empty string if the index is out of scope. HierarchyN is valid in rules only.

HierarchyN cannot be used in TurboIntegrator processes. The presence of this function in a process will prevent the process from compiling.

### Syntax

```
HierarchyN (DimName, index);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| index | A value less than or equal to the number of hierarchies in the dimension.<br>If this argument is less than 1, or greater than the number of hierarchies in the dimension, the function returns 0. |

#### Example

This example returns 'CustomerTarget', which is the third hierarchy in the model dimension.

```
HierarchyN ('model', 3);
```

# Logical Rules Functions

Logical functions to use in rules.

## CONTINUE

When included as part of a rules expression, CONTINUE allows a subsequent rule with the same area definition to be executed. Normally, Planning Analytics only executes the first rule encountered for a given area.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
CONTINUE
```

### Arguments

None.

#### Example

```
['Jan']= if(!region @= 'Argentina',10,CONTINUE);
```

```
['Jan']=20;
```

In this example, all cells identified by January and Argentina are assigned a value of 10. Cells identified by Jan and any other Region element are assigned a value of 20.

## IF

IF returns one value if a logical expression you specify is TRUE and another value if it is FALSE.

This function is valid in rules only.

TurboIntegrator uses its own IF function that is capable of evaluating multiple logical expressions.

### Syntax

```
IF(expression, true_value, false_value)
```

| Argument | Description |
|---|---|
| expression | Any value or expression that can be evaluated to TRUE or FALSE. |
| true_value | The value that is returned if expression is TRUE. |
| false_value | The value that is returned if expression is FALSE. |

#### Example

IF(1<2, 4, 5) returns 4.

IF(1>2, 'ABC', 'DEF') returns 'DEF'.

### STET

The STET function cancels the effect of a rule for a particular element.

This is a rules function, valid only in Planning Analytics rules. This function cannot be used in TurboIntegrator processes.

#### Syntax

```
STET
```

#### Arguments

None.

#### Example

In this example, the rule dictates that the value for Sales is always 100, except for the intersection of Sales and the element France from the Region dimension.

```
['Sales'] = IF(!Region @= 'France',STET, 100);
```

## Mathematical Rules Functions

Mathematical operators to use in rules.

### ABS

ABS returns the absolute value of a number.

This function is valid in both rules and TurboIntegrator processes.

#### Syntax

```
ABS(x)
```

| Argument | Description |
|---|---|
| x | The number for which you want to find the absolute value. |

#### Example

ABS(-1.2) returns 1.2

### ACOS

ACOS returns the angle, in radians, whose cosine is x.

This function is valid in both rules and TurboIntegrator processes.

#### Syntax

```
ACOS(x)
```

| Argument | Description |
|---|---|
| x | The cosine of the angle you want to find. x must be between -1 and 1; otherwise the function returns an error. |

**Example**

ACOS(0) returns 1.5708.

## ASIN

ASIN returns the angle, in radians, whose sine is x.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ASIN(x)
```

| Argument | Description |
|---|---|
| x | The sine of the angle you want to find. x must be between -1 and 1; otherwise the function returns an error. |

**Example**

ASIN(1) returns 1.5708.

## ATAN

ATAN returns the angle, in radians, whose tangent is x. The result is between -pi/2 and +pi/2.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ATAN(x)
```

| Argument | Description |
|---|---|
| x | The tangent of the angle you want to find. |

**Example**

ATAN(1) returns .7854.

## COS

COS returns the cosine of an angle expressed in radians.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
COS(x)
```

| Argument | Description |
|---|---|
| x | An angle, expressed in radians, for which you want to find the cosine. |

**Example**

COS(0) returns 1.

## EXP

EXP returns the natural anti-log of a number.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
EXP(x)
```

| Argument | Description |
|---|---|
| x | A number for which you want to find the natural anti-log. |

**Example**

EXP(1) returns 2.71828.

## INT

INT returns the largest integer that is less than or equal to a specified value.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
INT(x)
```

| Argument | Description |
|---|---|
| x | A numeric value. |

**Example**

INT(5.6) returns 5.

INT(-5.6) returns -6.

## ISUND

ISUND returns 1 if a specified value is undefined; otherwise it returns 0.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ISUND(x)
```

| Argument | Description |
|---|---|
| x | A number or expression. |

**Example**

ISUND(5.2) returns 0.

ISUND(1/0) returns 1.

## LN

LN returns the natural logarithm (base e) of a number.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
LN(x)
```

| Argument | Description |
|---|---|
| x | A positive number. The function returns an error if x is negative or zero. |

**Example**

LN(10) returns 2.302585093.

## LOG

LOG returns the base 10 logarithm of a positive number.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
LOG(x)
```

| Argument | Description |
|---|---|
| x | A positive number. The function returns an error if x is negative or zero. |

**Example**

LOG(10) returns 1.

## MAX

MAX returns the largest number in a pair of values.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
MAX(num1, num2)
```

| Argument | Description |
|---|---|
| num1 | The first in a pair of values. |
| num2 | The second in a pair of values. |

**Example**

MAX(10, 3) returns 10.

## MIN

MIN returns the smallest number in a pair of values.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
MIN(num1, num2)
```

| Argument | Description |
|---|---|
| num1 | The first in a pair of values. |
| num2 | The second in a pair of values. |

**Example**

MIN(10, 3) returns 3.

## MOD

MOD returns the remainder of dividing a number by a divisor.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
MOD(number, divisor)
```

| Argument | Description |
|---|---|
| number | The number for which you want to find the remainder. |
| divisor | The value by which the number argument is divided. |

**Example**

MOD(10, 3) returns 1.

## RAND

RAND generates a random number that is uniformly distributed between 0 and 1. The random number generator is seeded when Planning Analytics is loaded.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
RAND.
```

### Arguments

None.

#### Example

RAND generates a random number that is uniformly distributed between 0 and 1

## ROUND

ROUND rounds a given number to the nearest integer. Rounding can be done in a variety of ways.

This function is valid in both rules and TurboIntegrator processes.

The most basic form of rounding is to replace an arbitrary number by an integer. There are many ways of rounding a number y to an integer q.

The most common ones are:

- **Round to nearest**

  q is the integer that is closest to y (see "**Round away from zero**" for tie-breaking rules).
- **Round towards zero** (or truncate)

  q is the integer part of y, without its fraction digits.
- **Round down** (or take the floor)

  q is the largest integer that does not exceed y.
- **Round up** (or take the ceiling)

  q is the smallest integer that is not less than y.
- **Round away from zero**

  If y is an integer, q is y; else q is the integer that is closest to 0 and is such that y is between 0 and q.

TurboIntegrator essentially uses the **Round down** method of *floor(x + .5)*. Microsoft Excel uses the **Round to nearest** method. This can result in different integers depending on whether you are using a TurboIntegrator process or working in Excel.

### Syntax

```
ROUND(number)
```

| Argument | Description |
|---|---|
| number | The number you want to round. |

#### Example

ROUND(1.46) returns 1.

## ROUNDP

ROUNDP rounds a given number at a specified decimal precision.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
ROUNDP(number, decimal)
```

| Argument | Description |
|---|---|
| number | The number you want to round. |
| decimal | The decimal precision at which to apply the rounding. If this argument is positive, the function rounds the specified number of digits to the right of the decimal point. If this argument is negative, the function rounds the specified number of digits to the left of the decimal point.<br>The decimal argument must be between -15 and 15, inclusive. |

#### Example

ROUNDP(1.46, 1) returns 1.5.

ROUNDP(1.466, 2) returns 1.47.

ROUNDP(234.56, -1) returns 230.00.

ROUNDP(234.56, 0) returns 235.00.

## SIGN

SIGN determines if a number is positive, negative, or zero. The function returns 1 if the number is positive, -1 if the number is negative, and 0 if the number is zero.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
SIGN(number)
```

| Argument | Description |
|---|---|
| number | A number. |

#### Example

SIGN(-2.5) returns -1.

## SIN

SIN returns the sine of a given angle.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
SIN(x)
```

| Argument | Description |
|---|---|
| x | A value, expressed in radians, for which you want the sine. |

#### Example

SIN(1.5708) returns 1.

## SQRT

SQRT returns the square root of a given value.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
SQRT(x)
```

| Argument | Description |
|---|---|
| x | Any positive value. SQRT returns an error if x is negative. |

#### Example

SQRT(16) returns 4.

## TAN

TAN returns the tangent of a given angle.

This function is valid in both rules and TurboIntegrator processes.

### Syntax

```
TAN(x)
```

| Argument | Description |
|---|---|
| x | A value, expressed in radians, for which you want the tangent. |

#### Example

TAN(0) returns 0.

TAN(.7854) returns 1.

# Text Rules Functions

Rules to manage text in rules.

## CAPIT

CAPIT applies initial capitalization to every word in a string.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
CAPIT(string)
```

| Argument | Description |
|---|---|
| string | A text string. |

#### Example

CAPIT('first quarter sales') returns 'First Quarter Sales'.

## CHAR

CHAR returns the character identified by a given ASCII numeric code.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
CHAR(number)
```

| Argument | Description |
|---|---|
| number | An ASCII code number. This number must be between 1 and 255, inclusive. |

#### Example

CHAR(100) returns 'd'.

## CODE

CODE returns the ASCII numeric code for a specified character within a string.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
CODE(string, location)
```

| Argument | Description |
|---|---|
| string | A text string. |
| location | A number specifying the character within the string for which you want to find the ASCII code value. |

#### Example

CODE('321', 2) returns 50.

CODE('End', 3) returns 100.

## CODEW

CODEW returns the UTF-8 numeric code for a specified character within a string.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
CODEW(string, location)
```

| Argument | Description |
|---|---|
| string | A text string. |
| location | A number specifying the character within the string for which you want to find the UTF-8 code value. |

#### Example

CODEW('321', 2) returns 32.

CODEW('End', 3) returns 64.

## DELET

DELET returns the result of deleting a specified number of characters from a specified starting point within a string.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
DELET(string, start, number)
```

| Argument | Description |
|---|---|
| string | A text string. |
| start | The character at which to begin deletion. |
| number | The number of characters to delete. |

#### Example

DELET('payment', 3, 3) returns 'pant'.

## FILL

FILL repeats a given string as necessary to return a string of a specified length.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
FILL(string, length)
```

| Argument | Description |
|---|---|
| string | A text string. This string is repeated as necessary to achieve the specified length. |
| length | The length of the string you want the function to return. |

**Example**

`FILL('-', 5)` returns `'-----'`.

`FILL('ab', 5)` returns `'ababa'`.

## INSRT

INSRT inserts one string into another string at a specified insertion point.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
INSRT(string1, string2, location)
```

| Argument | Description |
|---|---|
| string1 | A text string. |
| string2 | A text string. |
| location | The position in string2 at which you want to insert string1. The function inserts string1 into string2 immediately prior to the character you specify. |

**Example**

INSRT('ABC', 'DEF', 2) returns DABCEF.

## LONG

LONG returns the length of a string.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
LONG(string)
```

| Argument | Description |
|---|---|
| string | A text string. |

**Example**

LONG('Sales') returns 5.

## LOWER

LOWER converts all upper case characters in a string to lower case.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
LOWER(string)
```

| Argument | Description |
|---|---|
| string | A text string. |

#### Example

LOWER('First Quarter Sales') returns 'first quarter sales'.

## NUMBR

NUMBR converts a string to a number. The string passed to the NUMBR function must use. (period) as the decimal separator and , (comma) as the thousand separator. Any other decimal/thousand separators will cause incorrect results.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
NUMBR(string)
```

| Argument | Description |
|---|---|
| string | The string you want to convert to a number. All characters other than '0' through '9', '+', '-', '.', and 'E' are ignored. |

#### Example

NUMBR('-5.6') returns -5.6.

NUMBR('-5A. B6C') returns -5.6.

## SCAN

SCAN returns a number indicating the starting location of the first occurrence of a specified substring within a string. If the substring does not occur in the given string, the function returns 0.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
SCAN(substring, string)
```

| Argument | Description |
|---|---|
| substring | The substring you are trying to locate. |

| Argument | Description |
|---|---|
| string | The string within which you are searching for the substring. |

The arguments to this function are case-sensitive. The capitalization used in the **substring** argument must exactly match the capitalization used in the **string** argument for the function to return a non-zero value.

#### Example

`SCAN('scribe', 'described')` returns 3.

However, `SCAN('Scribe', 'described')` returns 0, because the case in the **substring** argument (`Scribe`) does not match the case in the **string** argument (`described`).

## STR

STR converts a floating point number to a string representing the value in decimal notation.

The number passed to the STR function must use "." (period) as the decimal separator and "," (comma) as the thousand separator. Any other decimal or thousand separators will cause incorrect results.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
STR(number, length, decimal)
```

*Table 16. STR arguments*

| Argument | Description |
|---|---|
| number | The floating point number being converted to a string.<br>This number can contain a positive or negative sign.<br>This number can contain decimal places. |

| *Table 16. STR arguments (continued)* | |
|---|---|
| **Argument** | **Description** |
| length | The desired count of characters in the string representation, including sign, separators, decimal, or decimal places.<br><br>The **length** argument value should be a positive number greater than "0". If the **length** argument value is "0" or a negative number, the function returns an empty string.<br><br>If the count of digits in the number is less than the **length** argument value, the function inserts leading blank spaces to attain this length after inserting sign, separators, decimal, or decimal places.<br><br>If the count of digits in the whole number exceeds the **length** argument value **and** the **decimal** argument value is "0", the function truncates the whole number to attain this length.<br><br>If the count of digits in the number exceeds the **length** argument value **and** the **decimal** argument value is greater than "0", the function preserves the whole number and uses the specified decimal places. |
| decimal | The number of decimal places to include in the function result.<br><br>If this parameter is "0", a decimal point is not included.<br><br>If the number specified has more decimal places than the **decimal** argument, the function result is rounded. |

All arguments are required and you cannot pass empty argument values.

**Note:** There is a limitation when using STR with large floating point values. If the count of whole number digits in the **number** argument value exceeds the **length** argument value by more than 5, the function returns an empty string. For example, STR(14723017.2245, 4, 2) returns "14723017.22". The whole number portion of the **number** argument value has 8 digits, which is **not** more than 5 greater than the **length** argument value of 4 (8<5+4). However, STR(14723017.2245, 2, 2) returns an empty string, because the whole number portion of the **number** argument value has 8 digits, which **is** more than 5 greater than the **length** argument value of 2 (8>5+2)

**Examples**

| Function call | Number | Length | Decimal | Result |
|---|---|---|---|---|
| STR(3.14159, 6, 2) | 3.14159 | 6 | 2 | " 3.14" |
| STR(-3.14159, 6, 0) | -3.14159 | 6 | 0 | " -3" |
| STR(3.14159, 5, 3) | 3.14159 | 5 | 3 | "3.142" |
| STR(1000000, 4, 0) | 1000000 | 4 | 0 | "1000" |

| Function call | Number | Length | Decimal | Result |
|---|---|---|---|---|
| | | | | Note that the number is truncated. |
| STR(1000000, 4, 2) | 1000000 | 4 | 2 | "1000000.00"<br>Note that the number is not truncated because decimal is specified. |
| STR(10, 2, 4) | 10 | 2 | 4 | "10.0000" |
| STR(120536.74391, 8, 0) | 120536.74391 | 8 | 0 | " 120536"<br>The result includes left padding of two spaces to attain the specified length of 8. |
| STR(120536.74391, 5, 0) | 120536.74391 | 5 | 5 | "12053"<br>The count of digits in the whole number exceeds the **length** argument value **and** the **decimal** argument value is "0", so the function truncates the whole number to attain the specified length of 5. |

## SUBST

SUBST returns a substring of a given string.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
SUBST(string, beginning, length)
```

| Argument | Description |
|---|---|
| string | The string from which you want to extract the substring. |
| beginning | The character at which the substring begins. |
| length | The length of the substring. |

**Example**

SUBST('Retirement', 3, 4) returns 'tire'.

### TRIM

TRIM returns the result of trimming any leading and trailing blanks from a string.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
TRIM(string)
```

| Argument | Description |
|---|---|
| string | A text string. |

**Example**

TRIM(' First Quarter ') returns 'First Quarter'.

### UPPER

UPPER converts a text string to upper case.

This function is valid in both TM1 rules and TurboIntegrator processes.

### Syntax

```
UPPER(string)
```

| Argument | Description |
|---|---|
| string | A text string. |

**Example**

UPPER('First Quarter Results') returns FIRST QUARTER RESULTS.

## Miscellaneous Rules Functions

Rules functions not found in other categories.

### FEEDERS

When you use a SKIPCHECK declaration to restore the sparse consolidation in a TM1 rule, you must also ensure that all rules-derived cells are identified by feeder statements. To do this, insert a FEEDERS declaration immediately following all rules statements:

```
FEEDERS;
```

Immediately following the FEEDERS declaration you should create feeders statements that identify the rules-derived cells in the cube.

For a complete discussion of TM1 rules, including sparse consolidation and the creation of feeders, please refer to *TM1 Rules*.

### FEEDSTRINGS

Rule-generated string values are not displayed when a view is zero-suppressed unless the string resides in a cell that is fed. To enable feeding of string cells, insert the FEEDSTRINGS declaration as the first line of your rule.

```
FEEDSTRINGS;
```

Once this declaration is in place, you can set up feeders for string cells in a cube view, and rely on the string to be available to other rules even if the view is zero-suppressed. Statements that define feeders for string cells should be created following the FEEDERS declaration in your rule.

As in the case of numeric feeders, a feed to a consolidated cell results in feeding of all components of the consolidation. Because you can store strings in consolidated cells, you must pay special attention if such cells are used to feed other cells. Overuse of string feeders can result in calculation explosions and poor application performance.

For a complete discussion of TM1 rules, including the creation of feeders, please refer to *TM1 Rules*.

### SKIPCHECK

You can restore sparse consolidation and improve performance by inserting a SKIPCHECK declaration at the beginning of the TM1 rule.

During consolidations, TM1 uses a sparse consolidation algorithm to skip over cells that contain zero or are empty. This algorithm speeds up consolidation calculations in cubes that are highly sparse. A sparse cube is a cube in which the number of populated cells as a percentage of total cells is low.

When consolidating data in cubes that have rules defined, TM1 turns off this sparse consolidation algorithm because one or more empty cells may in fact be calculated by a rule. (Skipping rules-calculated cells will cause consolidated totals to be incorrect). When the sparse consolidation algorithm is turned off, every cell is checked for a value during consolidation. This can slow down calculations in cubes that are very large and sparse.

```
SKIPCHECK;
```

If your rule uses a FEEDSTRINGS statement, the SKIPCHECK statement should be the second statement in your rule. If your rule does not use a FEEDSTRINGS statement, the SKIPCHECK statement should be the first statement in your rule.

When you use SKIPCHECK to restore sparse consolidation, you must also ensure that your rule includes a FEEDERS declaration and that all rules-derived cells are identified by feeder statements.

For a complete discussion of TM1 rules, including sparse consolidation and the creation of feeders, please refer to *TM1 Rules*.

## TurboIntegrator Functions

TurboIntegrator lets you manipulate Planning Analytics data and metadata when you define a process.

This is accomplished through the use of functions in the Prolog, Metadata, Data, and Epilog sub-tabs within the Advanced tab of the TurboIntegrator window. These sub-tabs include generated statements based on settings and options you select when defining a TurboIntegrator process. Any functions you create must appear after the generated statements. For details on creating processes with TurboIntegrator, see the *TurboIntegrator* documentation.

The TurboIntegrator functions in this section are sorted by category.

String arguments to TurboIntegrator functions must be enclosed in single quotation marks. A semi-colon (;) must be included to indicate the end of each function in the TurboIntegrator window.

In addition to these TurboIntegrator functions, you can also incorporate all standard Planning Analytics Rules functions in a process definition, with the exception of the STET function.

Each argument to TurboIntegrator functions is limited to 256 bytes. A TurboIntegrator function can accept multiple arguments, and each argument is limited to 256 bytes.

## TurboIntegrator reserved words

To prevent errors in your TurboIntegrator scripts, you should avoid creating variables with names that match any of the words listed in the following categories.

There are four categories of reserved words in TurboIntegrator:

- Rules function names - See “Rules functions” on page 971 for a complete listing of all rules function names.
- TurboIntegrator function names - See “TurboIntegrator Functions” on page 1036 for a complete listing of all TurboIntegrator function names.
- Implicit local variable names - See “TurboIntegrator Local Variables” on page 1237 for a complete listing of all TurboIntegrator implicit local variables names.
- TurboIntegrator keywords - These keywords are reserved and should not be used as variables in your scripts:
  - break
  - else
  - elseif
  - end
  - endif
  - if
  - while
- Process presentation keywords - These keywords are used to manage the presentation of scripts in the process editor.
  - #Section

## ASCII and Text TurboIntegrator Functions

These functions pertain to ASCII and Text.

### ASCIIDelete

ASCIIDelete deletes an ASCII file.

This function is valid in TM1 TurboIntegrator processes only.

#### Syntax

```
ASCIIDelete(FileName);
```

| Argument | Description |
|---|---|
| FileName | The name of the ASCII file you want to delete. If a full path is not specified, TM1 searches for the file in the server data directory. |

**Example**

This example deletes the ASCII file named 2002Q1Results.cma from the `C:\exported_data` directory.

```
ASCIIDelete('C:\exported_data\2002Q1Results.cma');
```

## ASCIIOutput

ASCIIOutput writes a comma-delimited record to an ASCII file.

This function is valid in TM1 TurboIntegrator processes only.

The ASCII file is opened when the first record is written, and is closed when the TurboIntegrator procedure (Prolog, Metadata, Data, or Epilog) containing the ASCIIIOutput function finishes processing.

Each output record generated by ASCIIOutput is limited to 64 kilobytes. If an output record exceeds 64 kilobytes, the record is truncated and a warning is logged in the TM1ProcessError.log file.

When ASCIIOutput encounters a String argument that pushes the output record beyond the 64 kilobyte limit, it ignores that argument and any further arguments. For example, if there are 10 String arguments and output for the first seven arguments total 65,500 bytes (just under the 64 KB limit) while the output for the eighth argument is 50 bytes, only the output for the first seven arguments will be written to the record, as the eighth argument causes the output to exceed the 64 kilobyte limit. If there are ten String arguments and the first argument is over 64 kilobytes, no output is written to the record.

If you use the ASCIIOutput function to write to the same file in multiple procedures (tabs) of a TurboIntegrator process, the file will be overwritten each time it is opened for a new procedure.

The ASCIIOutput function generates a minor error if an error occurs while writing the ASCII file. In addition, the function returns a value upon execution: 1 if the function successfully writes the ASCII file and 0 on failure.

**Note:** The error will be generated and the value returned only when ASCIIOutput is writing to a disk other than the one that the server is running on. For example, if the server is running on the C: drive and ASCIIOutput is writing to the F: drive, and the F: drive runs out of space, the error will be trapped and the server remains alive. If the server is running on the C: drive while ASCIIOutput is also writing to the C: drive, and that drive runs out of space, the server will terminate (as expected).

**Note:** The ability to execute the ASCIIOutput function when the data source is a cube view is determined by the **Allow Export as Text** capability assignment, which is set per user group. If a user is a member of a group which is denied the ability to export data as text, any attempt by the user to execute ASCIIOutput results in the process exiting with a permission error. The process message log indicates "`Execution was aborted. No security access for ASCIIOutput.`"

For details on how the **Allow Export as Text** capability is set, see "Capability Assignments" in *TM1 Operations*.

**Note:** The ASCIIOutput function places the 0x1A hexadecimal character at the end of all generated files. However, TM1 Web cannot open a Websheet that contains the 0x1A hexadecimal character.

If you use ASCIIOutput to export TM1 data to an ASCII file and then attempt to open the file in a TM1 Websheet, you will encounter the following error.

Error occurred while converting the MS Excel workbook into XML format, hexadecimal value 0x1A is an invalid character.

If you remove the 0x1A hexadecimal character from the Websheet, the file will open in TM1 Web.

## Syntax

```
ASCIIOutput(FileName, String1, String2, ...String n);
```

| Argument | Description |
|---|---|
| FileName | A full path to the ASCII file to which you want to write the record. Path must include a file extension. |

| Argument | Description |
|---|---|
| String1...String*n* | A string that corresponds to each field you want to create in the ASCII file. This argument can be a string or a TurboIntegrator variable for a string. |

**Example**

This example writes a record to the NewCube.cma ASCII file. Each field in the record corresponds to a variable assigned by TurboIntegrator to a column in your data source.

```
ASCIIOutput('NewCube.cma', V1, V2, V3, V4, V5 );
```

## ASCIIOutputOpen

ASCIIOutputOpen appends or overwrites content in a specified existing file.

This function is valid in TurboIntegrator processes only.

## Syntax

```
ASCIIOutputOpen(FileName, OpeningMode);
```

| Argument | Description |
|---|---|
| FileName | A full path or URL to the ASCII file to which you want to write content. The file name must include a file extension., either .txt or .csv. |
| OpeningMode | A bit field flag used to configure the file opening mode. The first bit (0x1 or 0b00000001), if set, indicates the file is opened in append mode. If not set, the existing file is opened in overwrite mode.<br>The second bit (0x2 or 0b00000010), if set, opens the file in FILE_SHARE_READ mode on Windows. The second bit has no effect on Linux.<br>FILE_OPEN_APPEND( ) and FILE_OPEN_SHARED( ) are two helper functions that can be used with ASCIIOutputOpen. You can use them in place of the bit field flag.<br>FILE_OPEN_APPEND( ) returns 0x1. Specify this function as the OpeningMode value to open the file in append mode. Omit the function to open the file in overwrite mode.<br>FILE_OPEN_SHARED( ) returns 0x2. Use this in conjunction with FILE_OPEN_APPEND( ) to open the file in FILE_SHARE_READ mode on Windows. |

**Examples**

Opens the foo.txt file in overwrite mode, without shared read access:

```
ASCIIOutputOpen('foo.txt', 0);
```

Opens the foo.txt file in append mode, without shared read access:

```
ASCIIOutputOpen('foo.txt', 1);
```

Opens the foo.txt file in overwrite mode, shared read access enabled:

```
ASCIIOutputOpen('foo.txt', 2);
```

Opens the foo.txt file, in append mode, shared read access enabled:

```
ASCIIOutputOpen('foo.txt', 3);
```

Opens the foo.txt file in append mode, without shared read access:

```
ASCIIOutputOpen('foo.txt', FILE_OPEN_APPEND());
```

Opens the foo.txt file in append mode, shared read access enabled:

```
ASCIIOutputOpen('foo.txt', FILE_OPEN_APPEND()+FILE_OPEN_SHARED());
```

## NumberToString

NumberToString converts a number to a string, using the decimal separator for the current user locale.

This function is valid in TM1 TurboIntegrator processes only.

In Microsoft Windows, the decimal separator is a Regional Options setting.

The output of this function is similar to the 'general' number format; it does not use thousands separators and uses the minus sign (-) to denote negative numbers.

### Syntax

```
NumberToString(Value);
```

| Argument | Description |
|---|---|
| Value | The real value that you want to convert to a string. |

#### Example

```
nRET = NumberToString(1234.5);
```

## NumberToStringEx

NumberToStringEx converts a number to a string, using the passed string format, decimal separator, and thousands separator.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
NumberToStringEx(Value, NumericFormat, DecimalSep, ThousandsSep);
```

| Argument | Description |
|---|---|
| Value | The real value that you want to convert to a string. |

| Argument | Description |
|---|---|
| FormatString | A TM1 numeric format string that defines the format for the function output. Numeric formats are described inIBM Cognos *TM1 Perspectives, TM1 Architect, and TM1 Web* documentation. |
| DecimalSep | The decimal separator to be used in the output string. |
| ThousandsSep | The thousands separator to be used in the output string. |

**Example**

```
sRet=NUMBERTOSTRINGEX(7895.23,'#,0.##########', ',','.');
```

```
ASCIIOUTPUT('number_to_string.txt',sRet);
```

Will return in ascii file;

7.895,23

## SetInputCharacterSet

SetInputCharacterSet function lets you specify the character set used in a TurboIntegrator data source.

This function is valid in TM1 TurboIntegrator processes only.

When a TurboIntegrator process reads an external file as input, it needs to know the character set in which that external file was written. If the file contains a valid byte-order-mark, TM1 functions will correctly convert the file to UTF-8 if required.

For formats lacking a valid byte-order-mark, the characters must be converted from some other encoding to UTF-8. If the proper converters are present on the machine hosting the server, the input file will be converted to the Unicode character set required by TM1.

### Syntax

```
SetInputCharacterSet (CharacterSet);
```

| Argument | Description |
|---|---|
| CharacterSet | The character encoding in the input file to be used by the TurboIntegrator process.<br>If the CharacterSet argument is not a known character type, the type defaults to the system locale. |

These are the valid values for `CharacterSet`.

| Character Encoding | System Locale |
|---|---|
| TM1CS_ISO_8859_1 | ISO-8859-1 Latin-1, Western Europe |
| TM1CS_ISO_8859_2 | ISO-8859-2 Latin-2, Central Europe |

| Character Encoding | System Locale |
|---|---|
| TM1CS_ISO_8859_3 | ISO-8859-3 Latin-3, South Europe |
| TM1CS_ISO_8859_4 | ISO-8859-4 Latin-4, North Europe |
| TM1CS_ISO_8859_5 | ISO-8859-5 Latin/Cyrillic |
| TM1CS_ISO_8859_6 | ISO-8859-6 Latin/Arabic |
| TM1CS_ISO_8859_7 | ISO-8859-7 Latin/Greek |
| TM1CS_ISO_8859_8 | ISO-8859-8 Latin/Hebrew |
| TM1CS_ISO_8859_9 | ISO-8859-9 Latin-5, Turkish |
| TM1CS_ISO_8859_10 | ISO-8859-10 Latin-6, Nordic, |
| TM1CS_ISO_8859_11 | ISO-8859-11 Latin/Thai |
| TM1CS_ISO_8859_13 | ISO-8859-13 Latin-7, Baltic Rim |
| TM1CS_ISO_8859_14 | ISO-8859-14 Latin-8, Celtic |
| TM1CS_ISO_8859_15 | ISO-8859-15 Latin-9, replaces ISO-8859-1 |
| TM1CS_ISO_8859_16 | ISO-8859-16 Latin-10, South-Eastern Europe |
| TM1CS_WCP1250 | Microsoft Windows Central Europe |
| TM1CS_WCP1251 | Windows Cyrillic |
| TM1CS_WCP1252 | Windows Latin-1 multilingual |
| TM1CS_WCP1253 | Windows Greek |
| TM1CS_WCP1254 | Windows Turkish |
| TM1CS_WCP1255 | Windows Hebrew |
| TM1CS_WCP1256 | Windows Arabic |
| TM1CS_WCP1257 | Windows Baltic |
| TM1CS_WCP1258 | Windows Vietnam |
| TM1CS_WCP874 | Windows Thai |
| TM1CS_WCP932 | Windows Japanese |
| TM1CS_WCP936 | Windows Simplified Chinese |
| TM1CS_WCP949 | Windows Korean |

| Character Encoding | System Locale |
|---|---|
| TM1CS_WCP950 | Windows Traditional Chinese |
| TM1CS_KOI8R | Russian and Cyrillic (KOI8-R) |
| TM1CS_GB18030 | PRC version UNICODE |
| TM1CS_BIG5 | Traditional Chinese |
| TM1CS_SHIFTJIS | JIS 0201 + JIS 0208, slightly different from CP932 |
| TM1CS_SJIS0213 | JIS 0213-2004, non-BMP required. |
| TM1CS_EUC_JP | EUC Japanese |
| TM1CS_EUC_CN | EUC Simplified Chinese |
| TM1CS_EUC_KR | EUC Korean |
| TM1CS_UTF8 | UTF-8 |
| TM1CS_UTF16 | UTF-16 Little Endian |
| TM1CS_UTF16ESC | UNICODE notation |
| TM1CS_UTF32 | UTF-32 Little Endian |
| TM1CS_OS_DEFAULT | operating system default |
| TM1CS_LOCALPATH | local encoding but UNICODE notation on non-native. |

**Example**

```
SetInputCharacterSet ('TM1CS_ISO_8859_11');
```

This example specifies that the input character set for the TurboIntegrator data source is ISO-8859-11 Latin/Thai.

## SetOutputCharacterSet

SetOutputCharacterSet lets you specify the character set to be used when writing to a text file using TextOutput in a TurboIntegrator process.

This function is valid in TurboIntegrator processes only.

SetOutputCharacterSet should precede the TextOutput function in the process procedure (Prolog, Metadata, Data. Epilog) where the TextOutput function appears. For example, if you want to write to a text file using TextOutput in the Data procedure, you should first use SetOutputCharacterSet to specify the character set in the Data procedure.

### Syntax

```
SetOutputCharacterSet( FileName, CharacterSet );
```

| Argument | Description |
|---|---|
| FileName | A full path to the text file for which you want to specify a character set. The path must include a file extension.<br>This argument should be identical to the FileName argument for the TextOutput function. |
| CharacterSet | The character encoding to use when writing to the output file.<br>For more information on the valid values for CharacterSet, see “SetInputCharacterSet” on page 1041. |

## SetOutputEscapeDoubleQuote

SetOutputEscapeDoubleQuote allows you to escape double quotes that appear in element names or data values when exporting a cube view to a .csv file.

This function is valid in TM1 TurboIntegrator processes only.

When SetOutputEscapeDoubleQuote is included in your TurboIntegrator script and set to 1, the exported file retains the double quote positions as they appear in your source cube view by escaping each double quote within another pair of double quotes. For example, if an element in your source view is named "Region", the element is exported as """Region""" in the .csv output file.

When SetOutputEscapeDoubleQuote is *not* included in your TurboIntegrator script or is set to 0, the exported file does not escape any double quotes that appear in your source cube.

SetOutputEscapeDoubleQuote is used in conjunction with the ASCIIOutput function, which is the function that actually writes the output file. SetOutputEscapeDoubleQuote should precede ASCIIOutput in your TurboIntegrator script, and both functions should use the same FileName parameter value.

### Syntax

```
SetOutputEscapeDoubleQuote(FileName, Num);
```

| Argument | Description |
|---|---|
| FileName | A full path to the file to which you want to write the cube view. Path must include a file extension. |
| Num | A flag that determines if double quotes are escaped in the output file.<br>1 indicates that double quotes will be escaped in the output file.<br>0 indicates that double quotes will not be escaped in the output file. |

#### Example

```
SetOutputEscapeDoubleQuote('C:\temp\cube1.csv', 1);
```

This example escapes any double quotes encountered in the source cube view when writing output to the C:\temp\cube1.csv file.

## StringToNumber

StringToNumber converts a string to a number, using the decimal separator for the current user locale. If the input string is an invalid number string, the value returned will be an invalid floating point value. In Microsoft Windows, the decimal separator is a Regional Options setting.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
StringToNumber(String);
```

| Argument | Description |
|---|---|
| String | The string you want to convert to a number. |

#### Example

```
nRET = StringToNumber('123.45');
```

## StringToNumberEx

StringToNumberEx converts a string to a number, using the passed decimal separator and thousands separator. If the input string is an invalid number string, the value returned will be an invalid floating point value.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
StringToNumberEx(String, DecimalSep, ThousandsSep);
```

| Argument | Description |
|---|---|
| String | The string that you want to convert to a number. |
| DecimalSep | The decimal separator to be used in the output number. |
| ThousandsSep | The thousands separator to be used in the output number. |

#### Example

```
nRET = StringToNumberEx('12453.45', ' . ', ' , ');
```

## TextOutput

TextOutput writes a comma-delimited record to a text file.

This function is valid in TM1 TurboIntegrator processes only.

By default TextOutput writes characters in the locale character set of the server machine. To create a file in a different character set, call the function SetOutputCharacterSet before calling TextOutput.

The text file is opened when the first record is written, and is closed when the TurboIntegrator procedure (Prolog, Metadata, Data, or Epilog) containing the TextOutput function finishes processing.

If you use the TextOutput function to write to the same file in multiple procedures (tabs) of a TurboIntegrator process, the file will be overwritten each time it is opened for a new procedure.

Each output record generated by TextOutput is limited to 8000 bytes. If an output record exceeds 8000 bytes, the record is truncated and a warning is logged in the TM1ProcessError.log file.

When TextOutput encounters a String argument that pushes the output record beyond the 8000 byte limit, it ignores that argument and any further arguments. For example, if there are 10 String arguments and output for the first seven arguments total 7950 bytes while the output for the eighth argument is 51 bytes, only the output for the first seven arguments will be written to the record. If there are ten String arguments and the first argument is over 8000 bytes, no output will be written to the record.

The TextOutput function generates a minor error if an error occurs while writing the text file. In addition, the function returns a value upon execution: 1 if the function successfully writes the text file and 0 on failure.

The error will be generated and the value returned only when TextOutput is writing to a disk other than the one that the server is running on. For example, if the server is running on the C: drive and TextOutput is writing to the F: drive, and the F: drive runs out of space, the error will be trapped and the server remains alive. If the server is running on the C: drive while TextOutput is also writing to the C: drive, and that drive runs out of space, the server will terminate (as expected).

**Note:** The ability to execute the TextOutput function when the data source is a cube view is determined by the **Allow Export as Text** capability assignment, which is set per user group. If a user is a member of a group which is denied the ability to export data as text, any attempt by the user to execute TextOutput results in the process exiting with a permission error. The process message log indicates "`Execution was aborted. No security access for TextOutput.`"

For details on how the **Allow Export as Text** capability is set, see "Capability Assignments" in the *IBM Cognos TM1 Operations* documentation.

### Syntax

```
TextOutput(FileName, String1, String2, ...String n);
```

| Argument | Description |
|---|---|
| FileName | A full path to the text file to which you want to write the record. Path must include a file extension. |
| String1...String*n* | A string that corresponds to each field you want to create in the text file. This argument can be a string or a TurboIntegrator variable for a string. |

#### Example

```
TextOutput('NewCube.cma', V1, V2, V3, V4, V5 );
```

This example writes a record to the NewCube.cma file. Each field in the record corresponds to a variable assigned by TurboIntegrator to a column in your data source.

## Attribute Manipulation TurboIntegrator Functions

These functions facilitate the manipulation of attributes.

### ATTRNL

ATTRNL returns a numeric attribute for a specified element of a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ATTRNL(DimName, ElName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| ElName | An element of the dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the element. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` locale. If the attribute value for `fr` is not available, the base attribute value is returned |

### Example

In this example, the function returns the numeric value of the Engine Size attribute of the L Series 1.8L Sedan element in the Model dimension for the French locale.

```
ATTRNL('Model', 'L Series 1.8L Sedan', 'Engine Size', 'fr');
```

## ATTRSL

AttrSL returns a string attribute for a specified element of a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AttrSL(DimName, ElName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| ElName | An element of the dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the element. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the string value of the Currency attribute of the 10100 element in the Plan_Business_Unit dimension for the French locale.

```
AttrSL('Plan_Business_Unit', '10100', 'Currency', 'fr');
```

## AttrDelete

AttrDelete deletes an element attribute from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AttrDelete(DimName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | The dimension for which you want to delete an element attribute. |

| Argument | Description |
|---|---|
| AttrName | The name of the attribute you want to delete. |

**Example**

This example deletes the InteriorColor element attribute for the Model dimension.

```
AttrDelete('Model', 'InteriorColor');
```

## AttrInsert

AttrInsert creates a new element attribute for a dimension. The function can create a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AttrInsert(DimName, PrevAttr, AttrName, Type);
```

| Argument | Description |
|---|---|
| DimName | The dimension for which you want to create an element attribute. |
| PrevAttr | The attribute that precedes the attribute you are creating. |
| AttrName | The name you want to assign to the new attribute. |
| Type | The type of attribute. There are three possible values for the Type argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

**Example**

This example creates the InteriorColor string attribute for the Model dimension. This attribute is inserted after the Transmission attribute.

```
AttrInsert('Model', 'Transmission', 'InteriorColor','S');
```

## AttrPutN

AttrPutN assigns a value to a numeric element attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AttrPutN( Value, DimName, ElName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| Value | The numeric value you want to assign to an element attribute. |
| DimName | The parent dimension of the element for which you want to assign an attribute value. |
| ElName | The element for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 2257993 to the ProdCode attribute of the S Series 1.8L Sedan in the Model dimension.

```
AttrPutN(2257993, 'Model', ' S Series 1.8L Sedan ','ProdCode');
```

## AttrPutS

AttrPutS assigns a value to a string element attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AttrPutS(Value, DimName, ElName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| Value | The value you want to assign to an element attribute. |
| DimName | The parent dimension of the element for which you want to assign an attribute value. |
| ElName | The element for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code to which the Value applies.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the string Beige to the InteriorColor attribute of the S Series 1.8L Sedan in the Model dimension.

```
AttrPutS('Beige', 'Model', 'S Series 1.8L Sedan', 'InteriorColor');
```

## ChoreAttrDelete

ChoreAttrDelete deletes a chore attribute from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreAttrDelete(AttrName);
```

| Argument | Description |
|---|---|
| AttrName | The name of the chore attribute you want to delete. |

**Example**

This example deletes the Description attribute for chores on your TM1 server.

```
ChoreAttrDelete('Description');
```

## ChoreAttrInsert

ChoreAttrInsert creates a new attribute for chores on your TM1 server. The function can create a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** If you update an existing chore attribute, you must first delete the existing attribute using the function ChoreAttrDelete. You can then use ChoreAttrInsert to recreate the attribute with your desired changes. If you attempt to update an existing attribute without first deleting it, the insert fails without a warning or error. The existing attribute remains unchanged; it is neither updated nor overwritten.

### Syntax

```
ChoreAttrInsert( PrevAttrName, NewAttrName, AttrType);
```

| Argument | Description |
|---|---|
| PrevAttrName | The attribute that precedes the attribute you are creating. If there is no previous attribute or you want the new attribute to be the first attribute for chores, leave this argument empty. |
| NewAttrName | The name you want to assign to the new chore attribute. |
| AttrType | The type of attribute. There are three possible values for the AttrType argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

**Example**

This example creates the Description string attribute for chores. This attribute is inserted after the Owner attribute.

```
ChoreAttrInsert('Owner', 'Description', 'S');
```

## ChoreAttrN

ChoreAttrN returns a numeric attribute for a specified chore.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreAttrN(ChoreName, AttrName);
```

| Argument | Description |
|---|---|
| ChoreName | A valid chore name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the chore. |

**Example**

In this example, the function returns the numeric value of the Division_Code attribute of the Import chore.

```
ChoreAttrN('Import', 'Division_Code');
```

## ChoreAttrNL

ChoreAttrNL returns an attribute's numeric value for a specified chore with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreAttrNL(ChoreName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| ChoreName | A valid chore name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the chore. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

#### Example

In this example, the function returns the numeric value of the Division_Code attribute of the Import chore for the French locale.

```
ChoreAttrNL('Import', 'Division_Code', 'fr');
```

## ChoreAttrPutN

ChoreAttrPutN assigns a value to a numeric chore attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreAttrPutN(NumericValue, ChoreName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| NumericValue | The value you want to assign to a chore attribute. |
| ChoreName | The chore for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 7161994 to the Division_Code attribute of the Import chore for the French language locale code.

```
ChoreAttrPutN(7161994, 'Import', 'Division_Code','fr');
```

## ChoreAttrPutS

ChoreAttrPutS assigns a value to a string chore attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreAttrPutS(String, ChoreName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| String | The string you want to assign to a chore attribute. |
| ChoreName | The chore for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

### Example

This example assigns the string value Ricci to the Owner attribute of the Import chore, for the French language locale code.

```
ChoreAttrPutS('Ricci', 'Import', 'Owner', 'fr');
```

## ChoreAttrS

ChoreAttrS returns a string attribute for a specified chore.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreAttrS(ChoreName, AttrName);
```

| Argument | Description |
|---|---|
| ChoreName | A valid chore name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the chore. |

### Example

In this example, the function returns the string value of the Owner attribute of the Exchange_Rate_Updates chore.

```
ChoreAttrS('Exchange_Rate_Updates', 'Owner');
```

## ChoreAttrSL

ChoreAttrSL returns a string attribute value for a specified chore with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreAttrSL(ChoreName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| ChoreName | A valid chore name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the chore. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

### Example

In this example, the function returns the string value of the Owner attribute of the Depreciate_Inventory chore for the French locale.

```
ChoreAttrSL('Depreciate_Inventory', 'Owner', 'fr');
```

## CubeAttrDelete

CubeAttrDelete deletes a cube attribute from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CubeAttrDelete(AttrName);
```

| Argument | Description |
|---|---|
| AttrName | The name of the cube attribute you want to delete. |

#### Example

This example deletes the Description attribute for cubes on your TM1 server.

```
CubeAttrDelete('Description');
```

## CubeAttrInsert

CubeAttrInsert creates a new attribute for cubes on your TM1 server. The function can create a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** If you update an existing cube attribute, you must first delete the existing attribute using the function CubeAttrDelete. You can then use CubeAttrInsert to recreate the attribute with your desired changes. If you attempt to update an existing attribute without first deleting it, the insert fails without a warning or error. The existing attribute remains unchanged; it is neither updated nor overwritten.

### Syntax

```
CubeAttrInsert( PrevAttrName, NewAttrName, AttrType);
```

| Argument | Description |
|---|---|
| PrevAttrName | The attribute that precedes the attribute you are creating. If there is no previous attribute or you want the new attribute to be the first attribute for cubes, leave this argument empty. |
| NewAttrName | The name you want to assign to the new cube attribute. |
| AttrType | The type of attribute. There are three possible values for the AttrType argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

#### Example

This example creates the Description string attribute for cubes. This attribute is inserted after the Owner attribute.

```
CubeAttrInsert('Owner', 'Description', 'S');
```

## CubeAttrPutN

CubeAttrPutN assigns a value to a numeric cube attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CubeAttrPutN(NumericValue, CubeName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| NumericValue | The value you want to assign to a cube attribute. |
| CubeName | The cube for which you want to assign an attribute value. |

| Argument | Description |
|---|---|
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 07161994 to the AccountingCode attribute of the Sales cube for the French language locale code.

```
CubeAttrPutN(07161994, 'Sales', 'AccountingCode','fr');
```

## CubeAttrPutS

CubeAttrPutS assigns a value to a string cube attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CubeAttrPutS(String, CubeName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| String | The string you want to assign to a cube attribute. |
| CubeName | The cube for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the string value Prototype to the Description attribute of the Sales cube for the French language locale code.

```
CubeAttrPutS('Prototype', 'Sales', 'Description','fr');
```

## CubeATTRNL

CubeATTRNL returns a numeric attribute value for a specified cube with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes.

### Syntax

```
CubeATTRNL(CubeName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| CubeName | A valid cube name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the cube. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

### Example

In this example, the function returns the numeric value of the Accounting_Code attribute of the Product cube for the French locale.

```
CubeATTRNL('Product', 'Accounting_Code', 'fr');
```

## CubeATTRSL

CubeATTRSL returns a string attribute value for a specified cube with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes.

### Syntax

```
CubeATTRSL(CubeName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| CubeName | A valid cube name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the cube. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

#### Example

In this example, the function returns the string value of the Owner attribute of the Product cube for the French locale.

```
CubeATTRSL('Product', 'Owner', 'fr');
```

## DimensionAttrDelete

DimensionAttrDelete deletes a dimension attribute from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionAttrDelete(AttrName);
```

| Argument | Description |
|---|---|
| AttrName | The name of the dimension attribute you want to delete. |

**Example**

This example deletes the Description attribute for dimensions on your TM1 server.

```
DimensionAttrDelete('Description');
```

## DimensionAttrInsert

DimensionAttrInsert creates a new attribute for dimensions on your TM1 server. The function can create a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** If you update an existing dimension attribute, you must first delete the existing attribute using the function DimensionAttrDelete. You can then use DimensionAttrInsert to recreate the attribute with your desired changes. If you attempt to update an existing attribute without first deleting it, the insert fails without a warning or error. The existing attribute remains unchanged; it is neither updated nor overwritten.

### Syntax

```
DimensionAttrInsert( PrevAttrName, NewAttrName, AttrType);
```

| Argument | Description |
|---|---|
| PrevAttrName | The attribute that precedes the attribute you are creating. If there is no previous attribute or you want the new attribute to be the first attribute for dimensions, leave this argument empty. |
| NewAttrName | The name you want to assign to the new dimension attribute. |
| AttrType | The type of attribute. There are three possible values for the AttrType argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

**Example**

This example creates the Description string attribute for dimensions. Because there is no PrevAttrName parameter, this attribute is inserted as the first attribute for dimensions on your TM1 server.

```
DimensionAttrInsert('', 'Description', 'S');
```

## DimensionAttrPutN

DimensionAttrPutN assigns a value to a numeric dimension attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionAttrPutN(NumericValue, DimensionName, AttrName, [LocalLangCode] );
```

| Argument | Description |
|---|---|
| NumericValue | The value you want to assign to a dimension attribute. |
| DimensionName | The dimension for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

#### Example

This example assigns the value 07161994 to the AccountingCode attribute of the Models dimension for the French language locale code.

```
DimensionAttrPutN(07161994, 'Models', 'AccountingCode','fr');
```

## DimensionAttrPutS

DimensionAttrPutS assigns a value to a string dimension attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionAttrPutS(String, DimensionName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| String | The string you want to assign to a dimension attribute. |
| DimensionName | The dimension for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the string value Prototype to the Description attribute of the Model dimension for the French language locale code.

```
DimensionAttrPutS('Prototype', 'Model', 'Description','fr');
```

## DimensionATTRNL

DimensionATTRNL returns a numeric attribute value for a specified dimension with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionATTRNL(DimName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the numeric value of the Accounting_Code attribute of the Plan_Business_Unit dimension for the French locale.

```
DimensionATTRNL('Plan_Business_Unit', 'Accounting_Code', 'fr');
```

## DimensionATTRSL

DimensionATTRSL returns a string attribute value for a specified dimension with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionATTRSL(DimName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the string value of the Manager attribute of the Plan_Business_Unit dimension for the French locale.

```
DimensionATTRSL('Plan_Business_Unit', 'Manager', 'fr');
```

## ElementATTRNL

ElementATTRNL returns a numeric attribute for a specified element of a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ElementATTRNL(DimName, HierName, ElName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | An element of the dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the element. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the numeric value of the Engine Size attribute of the L Series 1.8L Sedan element in the Model dimension for the French locale. This example applies to the 2015 hierarchy.

```
ATTRNL('Model', '2015', 'L Series 1.8L Sedan', 'Engine Size', 'fr');
```

## ElementATTRSL

ElementATTRSL returns a string attribute for a specified element of a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ElementATTRSL(DimName, HierName, ElName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | An element of the dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the element. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the string value of the Currency attribute of the 10100 element in the Plan_Business_Unit dimension for the French locale.

```
ElementATTRSL('Plan_Business_Unit', '10100', 'Currency', 'fr');
```

## ElementAttrPutN

ElementAttrPutN assigns a value to a numeric element attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ElementAttrPutN( Value, DimName, HierName, ElName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| Value | The numeric value you want to assign to an element attribute. |
| DimName | The parent dimension of the element for which you want to assign an attribute value. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | The element for which you want to assign an attribute value. |

| Argument | Description |
|---|---|
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 2257993 to the ProdCode attribute of the S Series 1.8L Sedan in the Automobile hierarchy of the Model dimension.

```
ElementAttrPutN(2257993, 'Model', 'Automobile', ' S Series 1.8L Sedan ','ProdCode');
```

## ElementAttrPutS

ElementAttrPutS assigns a value to a string element attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ElementAttrPutS(Value, DimName, HierName, ElName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| Value | The value you want to assign to an element attribute. |
| DimName | The parent dimension of the element for which you want to assign an attribute value. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | The element for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the Value applies.<br>Valid LangLocaleCode values correspond to the international language codes listed in the Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

### Example

This example assigns the string Beige to the InteriorColor attribute of the S Series 1.8L Sedan in the Automobile hierarchy of the Model dimension.

```
ElementAttrPutS('Beige', 'Model', 'Automobile', 'S Series 1.8L Sedan', 'InteriorColor');
```

## ElementAttrInsert

ElementAttrInsert creates a new element attribute for a dimension. The function can create a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ElementAttrInsert(DimName, HierName, PrevAttr, AttrName, Type);
```

| Argument | Description |
|---|---|
| DimName | The dimension for which you want to create an element attribute. |
| HierName | The name of the hierarchy within the dimension. |
| PrevAttr | The attribute that precedes the attribute you are creating. |
| AttrName | The name you want to assign to the new attribute. |
| Type | The type of attribute. There are three possible values for the Type argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

### Example

This example creates the InteriorColor string attribute in the Automobile hierarchy in the Model dimension. This attribute is inserted after the Transmission attribute.

```
ElementAttrInsert('Model', 'Automobile', 'Transmission', 'InteriorColor','S');
```

## ElementAttrDelete

ElementAttrDelete deletes an element attribute from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ElementAttrDelete(DimName, HierName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | The dimension for which you want to delete an element attribute. |
| HierName | The name of the hierarchy within the dimension. |
| AttrName | The name of the attribute you want to delete. |

**Example**

This example deletes the InteriorColor element attribute from the Automobile hierarchy in the Model dimension.

```
ElementAttrDelete('Model', 'Automobile', 'InteriorColor');
```

## HierarchyAttrPutN

HierarchyAttrPutN assigns a value to a numeric attribute in a specified hierarchy within a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyAttrPutN(NumericValue, DimensionName, HierName, AttrName, [LocalLangCode] );
```

| Argument | Description |
|---|---|
| NumericValue | The value you want to assign to a dimension attribute. |
| DimensionName | The dimension for which you want to assign an attribute value. |
| HierName | The name of the hierarchy within the dimension. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 07161994 to the AccountingCode attribute of the Models dimension for the French language locale code. This change is applied to the Receivables hierarchy in the Models dimension.

```
HierarchyAttrPutN(07161994, 'Models', 'Receivables', 'AccountingCode','fr');
```

## HierarchyAttrPutS

HierarchyAttrPutS assigns a value to a string attribute in a specified hierarchy within a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyAttrPutS(String, DimensionName, HierName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| String | The string you want to assign to a dimension attribute. |
| DimensionName | The dimension for which you want to assign an attribute value. |
| HierName | The name of the hierarchy within the dimension. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

#### Example

This example assigns the string value Prototype to the Description attribute of the Model dimension for the French language locale code. This change is applied to the Receivables hierarchy in the Model dimension.

```
HierarchyAttrPutS('Prototype', 'Model', 'Receivables', 'Description','fr');
```

## HierarchyATTRN

HierarchyATTRN returns a numeric attribute for a specified hierarchy within a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyATTRN(DimName, HierName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| HierName | The name of the hierarchy within the dimension. |

| Argument | Description |
|---|---|
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

**Example**

In this example, the function returns the numeric value of the Accounting_Code attribute of the Plan_Business_Unit dimension. This example applies to the Equipment hierarchy.

```
HierarchyATTRN('Plan_Business_Unit', 'Equipment', 'Accounting_Code');
```

## HierarchyATTRS

HierarchyATTRS returns a string attribute for a specified hierarchy within a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyATTRS(DimName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| HierName | The name of the hierarchy within the dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

**Example**

In this example, the function returns the string value of the Manager attribute of the Plan_Business_Unit dimension. This example applies to the Equipment hierarchy.

```
HierarchyATTRS('Plan_Business_Unit', 'Equipment', 'Manager');
```

## HierarchyATTRNL

HierarchyATTRNL returns a numeric attribute value for a specified hierarchy within a dimension with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyATTRNL(DimName, HierName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |

| Argument | Description |
|---|---|
| HierName | The name of the hierarchy within the dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the numeric value of the Accounting_Code attribute of the Plan_Business_Unit dimension for the French locale. This function applies to the Equipment hierarchy.

```
HierarchyATTRNL('Plan_Business_Unit', 'Equipment', 'Accounting_Code', 'fr');
```

## HierarchyATTRSL

HierarchyATTRSL returns a string attribute value for a specified hierarchy within a dimension with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyATTRSL(DimName, HierName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |

| Argument | Description |
|---|---|
| HierName | The name of the hierarchy within the dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the string value of the Manager attribute of the Plan_Business_Unit dimension for the French locale. This function applies to the Equipment hierarchy.

```
HierarchyATTRSL('Plan_Business_Unit', 'Equipment', 'Manager', 'fr');
```

## HierarchySubsetATTRS

HierarchySubsetATTRS returns a string attribute for a specified subset associated with a hierarchy in a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetATTRS(DimName, HierName, SubName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |

| Argument | Description |
|---|---|
| HierName | The name of a hierarchy in a dimension. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

**Example**

In this example, the function returns the string value of the Manager attribute of the Sales subset from Europe hierarchy in the Plan_Business_Unit dimension.

```
HierarchySubsetATTRS('Plan_Business_Unit', 'Europe', 'Sales', 'Manager');
```

## HierarchySubsetATTRN

HierarchySubsetATTRN returns a numeric attribute for a specified subset associated with a hierarchy in a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetATTRN(DimName, HierName, SubName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| HierName | The name of a hierarchy in a dimension. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

**Example**

In this example, the function returns the numeric value of the Accounting_Code attribute of the Sales subset from the Europe hierarchy in the Plan_Business_Unit dimension.

```
HierarchySubsetATTRN('Plan_Business_Unit', 'Europe', 'Sales', 'Accounting_Code');
```

## HierarchySubsetATTRSL

HierarchySubsetATTRSL returns an attribute's string value for a specified subset (and locale) associated with a hierarchy in a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetATTRSL(DimName, HierName, SubName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| HierName | The name of a hierarchy in a dimension. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

#### Example

In this example, the function returns the string value of the Manager attribute of the Sales subset (from the Europe hierarchy) for the French locale.

```
HierarchySubsetATTRSL('Plan_Business_Unit', 'Europe', 'Sales', 'Manager', 'fr');
```

## HierarchySubsetATTRNL

HierarchySubsetATTRNL returns an attribute's numeric value for a specified subset (and locale) associated with a hierarchy in a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetATTRNL(DimName, HierName, SubName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| HierName | The name of a hierarchy in a dimension. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

#### Example

In this example, the function returns the numeric value of the Accounting_Code attribute of the Sales subset (from the Europe hierarchy) for the French locale.

```
HierarchySubsetATTRNL('Plan_Business_Unit', 'Europe', 'Sales', 'Accounting_Code', 'fr');
```

## HierarchySubsetAttrPutS

HierarchySubsetAttrPutS assigns a string value to an attribute for a specified subset associated with a hierarchy in a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetAttrPutS(String, DimName, HierName, SubName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| String | The string you want to assign to a dimension attribute. |
| DimName | The dimension for which you want to assign an attribute value. |
| HierName | The name of a hierarchy in a dimension. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

#### Example

This example assigns the string value Prototype to the Description attribute of the Z subset (from the 2016 hierarchy in the Model dimension) for the French language locale code.

```
HierarchySubsetAttrPutS('Prototype', 'Model', '2016', 'Z', 'Description','fr');
```

## HierarchySubsetAttrPutN

HierarchySubsetAttrPutN assigns a numeric value to an attribute for a specified subset associated with a hierarchy in a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetAttrPutN(NumericValue, DimName, HierName, SubName, AttrName, [LocalLangCode] );
```

| Argument | Description |
|---|---|
| NumericValue | The value you want to assign to a dimension attribute. |
| DimName | The dimension for which you want to assign an attribute value. |
| HierName | The name of a hierarchy in a dimension. |
| SubName | The name of a subset in a dimension. |

| Argument | Description |
|---|---|
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 07161994 to the AccountingCode attribute of the Z subset (from the 2016 hierarchy in the Models dimension) for the French language locale code.

```
HierarchySubsetAttrPutN(07161994, 'Models', '2016', 'Z', 'AccountingCode','fr');
```

## HierarchySubsetAttrInsert

HierarchySubsetAttrInsert creates a new attribute for subsets on your TM1 server. The function creates a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

**Note:** If you update an existing subset attribute, you must first delete the existing attribute using the function HierarchySubsetAttrDelete. You can then use HierarchySubsetAttrInsert to recreate the attribute with your desired changes. If you attempt to update an existing attribute without first deleting it, the insert fails without a warning or error. The existing attribute remains unchanged; it is neither updated nor overwritten.

```
HierarchySubsetAttrInsert( Dimension, Hierarchy, PrevAttrName, NewAttrName, AttrType);
```

| Argument | Description |
|---|---|
| Dimension | The name of the dimension whose subsets are being updated. |
| Hierarchy | The name of a hierarchy in a dimension. |
| PrevAttrName | The attribute that precedes the attribute you are creating. If there is no previous attribute or you want the new attribute to be the first attribute for subsets, leave this argument empty. |
| NewAttrName | The name you want to assign to the new subset attribute. |

| Argument | Description |
|---|---|
| AttrType | The type of attribute. There are three possible values for the AttrType argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

**Example**

This example creates the Description string attribute for subsets in the Z hierarchy of the Model dimension. Because there is no PrevAttrName parameter, this attribute is inserted as the first attribute for subsets on your TM1 server.

```
HierarchySubsetAttrInsert('Model', 'Z', '', 'Description', 'S');
```

## HierarchySubsetAttrDelete

HierarchySubsetAttrDelete deletes a subset attribute from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetAttrDelete(Dimension, Hierarchy, AttrName);
```

| Argument | Description |
|---|---|
| Dimension | The name of the dimension whose subset attribute is being deleted. |
| Hierarchy | The name of a hierarchy in a dimension. |
| AttrName | The name of the dimension attribute you want to delete. |

**Example**

This example deletes the Description attribute for subsets from the Z hierarchy in the Model dimension.

```
HierarchySubsetAttrDelete('Model', 'Z, 'Description');
```

## ProcessAttrDelete

ProcessAttrDelete deletes a process attribute from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessAttrDelete(AttrName);
```

| Argument | Description |
|---|---|
| AttrName | The name of the process attribute you want to delete. |

**Example**

This example deletes the Description attribute for processes on your TM1 server.

```
ProcessAttrDelete('Description');
```

## ProcessAttrInsert

ProcessAttrInsert creates a new attribute for processes on your TM1 server. The function can create a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** If you update an existing process attribute, you must first delete the existing attribute using the function ProcessAttrDelete. You can then use ProcessAttrInsert to recreate the attribute with your desired changes. If you attempt to update an existing attribute without first deleting it, the insert fails without a warning or error. The existing attribute remains unchanged; it is neither updated nor overwritten.

### Syntax

```
ProcessAttrInsert( PrevAttrName, NewAttrName, AttrType);
```

| Argument | Description |
|---|---|
| PrevAttrName | The attribute that precedes the attribute you are creating. If there is no previous attribute or you want the new attribute to be the first attribute for processes, leave this argument empty. |
| NewAttrName | The name you want to assign to the new process attribute. |
| AttrType | The type of attribute. There are three possible values for the AttrType argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

**Example**

This example creates the Description string attribute for processes. This attribute is inserted after the Owner attribute.

```
ProcessAttrInsert('Owner', 'Description', 'S');
```

## ProcessAttrN

ProcessAttrN returns a numeric attribute for a specified process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessAttrN(ProcessName, AttrName);
```

| Argument | Description |
|---|---|
| ProcessName | A valid process name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the process. |

#### Example

In this example, the function returns the numeric value of the Store_Code attribute of the Daily_Sales process.

```
ProcessAttrN('Daily_Sales', 'Store_Code');
```

## ProcessAttrNL

ProcessAttrNL returns an attribute's numeric value for a specified process with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessAttrNL(ProcessName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| ProcessName | A valid process name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the process. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the numeric value of the Store_Code attribute of the Daily_Sales process for the French locale.

```
ProcessAttrNL('Daily_Sales', 'Store_Code', 'fr');
```

## ProcessAttrPutN

ProcessAttrPutN assigns a value to a numeric process attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessAttrPutN(NumericValue, CubeName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| NumericValue | The value you want to assign to a process attribute. |
| ProcessName | The process for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 8051997 to the Store_Code attribute of the Daily_Sales process for the French language locale code.

```
ProcessAttrPutN(8051997, 'Daily_Sales', 'Store_Code','fr');
```

## ProcessAttrPutS

ProcessAttrPutS assigns a value to a string process attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessAttrPutS(String, ProcessName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| String | The string you want to assign to a process attribute. |
| ProcessName | The process for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the string value Ricci to the Owner attribute of the Import_Transactional process, for the French language locale code.

```
ProcessAttrPutS('Ricci', 'Import_transactional', 'Owner', 'fr');
```

## ProcessAttrS

ProcessAttrS returns a string attribute for a specified process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessAttrS(ProcessName, AttrName);
```

| Argument | Description |
|---|---|
| ProcessName | A valid process name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the process. |

#### Example

In this example, the function returns the string value of the Owner attribute of the Refresh_Cubes process.

```
ProcessAttrS('Refresh_Cubes', 'Owner');
```

## ProcessAttrSL

ProcessAttrSL returns a string attribute value for a specified process with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessAttrSL(ProcessName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| ProcessName | A valid process name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the process. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the string value of the Owner attribute of the Exchange_Rate_Update process for the French-Canada locale.

```
ProcessAttrSL('Exchange_Rate_Update', 'Owner', 'fr-CA');
```

## SubsetATTRS

SubsetATTRS returns a string attribute for a specified subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetATTRS(DimName, SubName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

**Example**

In this example, the function returns the string value of the Manager attribute of the Sales subset from the Plan_Business_Unit dimension.

```
SubsetATTRS('Plan_Business_Unit', 'Sales', 'Manager');
```

## SubsetATTRN

SubsetATTRN returns a numeric attribute for a specified subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetATTRN(DimName, SubName, AttrName);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

**Example**

In this example, the function returns the numeric value of the Accounting_Code attribute of the Sales subset from the Plan_Business_Unit dimension.

```
SubsetATTRN('Plan_Business_Unit', 'Sales', 'Accounting_Code');
```

## SubsetATTRSL

SubsetATTRSL returns an attribute's string value for a specified subset with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetATTRSL(DimName, SubName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the string value of the Manager attribute of the Sales subset for the French locale.

```
SubsetATTRSL('Plan_Business_Unit', 'Sales', 'Manager', 'fr');
```

## SubsetATTRNL

SubsetATTRNL returns an attribute's numeric value for a specified subset with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

## Syntax

```
SubsetATTRNL(DimName, SubName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| DimName | A valid dimension name. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the dimension. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

### Example

In this example, the function returns the numeric value of the Accounting_Code attribute of the Sales subset for the French locale.

```
SubsetATTRNL('Plan_Business_Unit', 'Sales', 'Accounting_Code', 'fr');
```

## SubsetAttrPutS

SubsetAttrPutS assigns a string value to an attribute for a specified subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetAttrPutS(String, DimensionName, SubName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| String | The string you want to assign to a dimension attribute. |
| DimensionName | The dimension for which you want to assign an attribute value. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute whose value you want to assign. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the string value Prototype to the Description attribute of the Z subset (from the Model dimension) for the French language locale code.

```
SubsetAttrPutS('Prototype', 'Model', 'Z', 'Description','fr');
```

## SubsetAttrPutN

SubsetAttrPutN assigns a numeric value to an attribute for a specified subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetAttrPutN(NumericValue, DimensionName, SubName, AttrName, [LocalLangCode] );
```

| Argument | Description |
|---|---|
| NumericValue | The value you want to assign to a dimension attribute. |
| DimensionName | The dimension for which you want to assign an attribute value. |
| SubName | The name of a subset in a dimension. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 07161994 to the AccountingCode attribute of the Z subset (from the Models dimension) for the French language locale code.

```
SubsetAttrPutN(07161994, 'Models', 'Z', 'AccountingCode','fr');
```

## SubsetAttrInsert

SubsetAttrInsert creates a new attribute for subsets on your TM1 server. The function creates a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** If you update an existing subset attribute, you must first delete the existing attribute using the function SubsetAttrDelete. You can then use SubsetAttrInsert to recreate the attribute with your desired changes. If you attempt to update an existing attribute without first deleting it, the insert fails without a warning or error. The existing attribute remains unchanged; it is neither updated nor overwritten.

### Syntax

```
SubsetAttrInsert( Dimension, PrevAttrName, NewAttrName, AttrType);
```

| Argument | Description |
|---|---|
| Dimension | The name of the dimension whose subsets are being updated. |
| PrevAttrName | The attribute that precedes the attribute you are creating. If there is no previous attribute or you want the new attribute to be the first attribute for subsets, leave this argument empty. |
| NewAttrName | The name you want to assign to the new subset attribute. |
| AttrType | The type of attribute. There are three possible values for the AttrType argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

#### Example

This example creates the Description string attribute for subsets in the Model dimension. Because there is no PrevAttrName parameter, this attribute is inserted as the first attribute for subsets on your TM1 server.

```
SubsetAttrInsert('Model', '', 'Description', 'S');
```

## SubsetAttrDelete

SubsetAttrDelete deletes a subset attribute from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetAttrDelete(Dimension, AttrName);
```

| Argument | Description |
|---|---|
| Dimension | The name of the dimension whose subset attribute is being deleted. |

| Argument | Description |
|---|---|
| AttrName | The name of the dimension attribute you want to delete. |

**Example**

This example deletes the Description attribute for subsets in the Model dimension.

```
SubsetAttrDelete('Model', 'Description');
```

## ViewAttrDelete

ViewAttrDelete deletes a view attribute for a specific cube from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewAttrDelete(CubeName, AttrName);
```

| Argument | Description |
|---|---|
| CubeName | The name of the cube whose view attribute is being deleted. |
| AttrName | The name of the view attribute you want to delete. |

**Example**

This example deletes the Description attribute for views of the Sales cube on your TM1 server.

```
ViewAttrDelete('Sales', 'Description');
```

## ViewAttrInsert

ViewAttrInsert creates a new attribute for views of a specific cube on your TM1 server. The function can create a string, numeric, or alias attribute.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** If you update an existing view attribute, you must first delete the existing attribute using the function ViewAttrDelete. You can then use ViewAttrInsert to recreate the attribute with your desired changes. If you attempt to update an existing attribute without first deleting it, the insert fails without a warning or error. The existing attribute remains unchanged; it is neither updated nor overwritten.

### Syntax

```
ViewAttrInsert( CubeName, PrevAttrName, NewAttrName, AttrType);
```

| Argument | Description |
|---|---|
| CubeName | The parent cube for which you want to insert a view attribute. |

| Argument | Description |
|---|---|
| PrevAttrName | The attribute that precedes the attribute you are creating. If there is no previous attribute or you want the new attribute to be the first attribute for views, leave this argument empty. |
| NewAttrName | The name you want to assign to the new view attribute. |
| AttrType | The type of attribute. There are three possible values for the AttrType argument:<br>• N - Creates a numeric attribute.<br>• S - Creates a string attribute.<br>• A - Creates an alias attribute. |

**Example**

This example creates the Description string attribute for views of the Sales cube. This attribute is inserted after the Owner attribute.

```
ViewAttrInsert('Sales', 'Owner', 'Description', 'S');
```

## ViewAttrN

ViewAttrN returns a numeric attribute for a specified view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewAttrN(CubeName, ViewName, AttrName);
```

| Argument | Description |
|---|---|
| CubeName | A valid cube name. |
| ViewName | A valid view name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the view. |

**Example**

In this example, the function returns the numeric value for the Category_Code attribute of the Product view of the Sales cube.

```
ViewAattrN('Sales', 'Product', 'Category_Code');
```

## ViewAttrNL

ViewAttrNL returns an attribute's numeric value for a specified view with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewAttrNL(CubeName, ViewName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| CubeName | The parent cube for the view whose attribute value you want to retrieve. |
| ViewName | A valid view name. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the view. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

#### Example

In this example, the function returns the numeric value for the Category_Code attribute of the Product view of the Sales cube, for the French locale.

```
ViewAttrNL('Sales', 'Product', 'Category_Code', 'fr');
```

## ViewAttrPutN

ViewAttrPutN assigns a value to a numeric view attribute.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewAttrPutN(NumericValue, CubeName, ViewName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| NumericValue | The value you want to assign to a view attribute. |
| CubeName | The parent cube of the view for which you want to assign an attribute value. |
| ViewName | The view for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the value 8222001 to the Category_Code attribute of the Product view of the Sales cube, for the French language locale code.

```
ViewAttrPutN(8222001, 'Sales', 'Product', 'Category_Code','fr');
```

## ViewAttrPutS

ViewAttrPutS assigns a string value to an attribute for a specified view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewAttrPutS(String, CubeName, ViewName, AttrName, [LangLocaleCode] );
```

| Argument | Description |
|---|---|
| String | The string you want to assign to a view attribute. |
| CubeName | The cube parent of the view for which you want to assign an attribute value. |
| ViewName | The name of the view for which you want to assign an attribute value. |
| AttrName | The attribute whose value you want to assign. |

| Argument | Description |
|---|---|
| LangLocaleCode | This optional parameter specifies the language locale code to which the NumericValue applies.<br>Valid LangLocaleCode values correspond to the ISO 639-1 international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the base attribute value is updated. |

**Example**

This example assigns the string value Rocheford to the Owner attribute of the Individual_Stores view of the Sales cube, for the French language locale code.

```
ViewAttrPutS('Rocheford', 'Sales', 'Individual_Stores', 'Owner','fr');
```

## ViewAttrS

ViewAttrS returns a string attribute for a specified view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewAttrS(CubeName, ViewName, AttrName);
```

| Argument | Description |
|---|---|
| CubeName | The parent cube of the view for which you want to return an attribute value. |
| ViewName | The view for which you want to return an attribute value. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the view. |

**Example**

```
ViewAttrS('Plan_Business_Unit', 'Sales', 'Manager');
```

In this example, the function returns the string value of the Manager attribute of the Sales view of the Plan_Business_Unit cube.

## ViewAttrSL

ViewAttrSL returns an attribute's string value for a specified view with respect to a given locale.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewAttrSL(CubeName, ViewName, AttrName, [LangLocaleCode]);
```

| Argument | Description |
|---|---|
| CubeName | The parent cube of the view for which you want to return an attribute value. |
| ViewName | The view for which you want to return an attribute value. |
| AttrName | The attribute for which you want to retrieve a value. This argument must be a valid attribute of the view. |
| LangLocaleCode | This optional parameter specifies the language locale code for which you want to return a value.<br>Valid LangLocaleCode values correspond to the international language codes listed in the }Cultures control dimension.<br>When the LangLocaleCode is not specified or is omitted, the user's current locale is used as the LangLocaleCode argument.<br>If an attribute value does not exist for the LangLocaleCode, the value for an associated parent LangLocaleCode is returned. If an attribute value does not exist for an associated parent LangLocaleCode, the base attribute value is returned.<br>For example if the LangLocaleCode is `fr-CA`, the function returns the attribute value for the `fr-CA` (French-Canada) locale if available. If the attribute value for `fr-CA` is not available, the function attempts to return the attribute value for the parent `fr` (French) locale. If the attribute value for `fr` is not available, the base attribute value is returned |

**Example**

In this example, the function returns the string value of the Manager attribute of the Sales view of the Plan_Business_Unit cube, for the French-Canada locale.

```
ViewAttrSL('Plan_Business_Unit', 'Sales', 'Manager', 'fr-CA');
```

# Chore Management TurboIntegrator Functions

These functions pertain to managing chores.

## ChoreError

ChoreError causes the immediate termination of a chore. It can be called from any process within a chore. The ChoreError TurboIntegrator function causes an immediate termination of a single chore. Chores terminated with this function are flagged with an error status.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreError;
```

### Arguments

None.

## ChoreQuit

ChoreQuit causes the immediate termination of a chore. It can be called from any process within a chore. The current chore is terminated with an error status, and a message is written to the server log file indicating that ChoreQuit was called to terminate the chore.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreQuit;
```

### Arguments

None.

## ChoreRollback

ChoreRollback initiates a chore rollback. When used inside a TurboIntegrator process, this function throws out all pending edits and cancels further processing. An error message appears in the tm1server.log and tm1processorerrorXXX.log files.

When used in a single-commit mode chore, **ChoreRollback** throws out all pending edits from all previous processes and chore execution stops with an error code. When used in a multi-commit mode chore, **ChoreRollback** throws out all pending edits from the current processes and chore execution stops with an error code. Changes that have already been committed cannot be rolled back.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ChoreRollback;
```

### Arguments

None.

## SetChoreVerboseMessages

SetChoreVerboseMessages is used to turn on (or off) more verbose reporting of messages to the Tm1s.log file. You can use this function to debug chores in which several processes call each other with the ExecuteProcess function.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
SetChoreVerboseMessages(Flag);
```

Passing a zero value turns off the output of these messages, passing a non-zero value enables the output of more verbose messages. By default this flag is off.

Use this function to turn on (or off) more verbose reporting of messages to the Tm1s.log file. This function is best used as an aid to debugging chores in which several processes call one another through use of the ExecuteProcess function.

Passing a zero value turns off the output of these messages, passing a non-zero value enables the output of more verbose messages. By default this flag is off.

| Argument | Description |
|---|---|
| Flag | Set to a non-zero value to enable more verbose messaging. Set to zero (default) to turn off verbose messaging. |

# Cube Manipulation TurboIntegrator Functions

These functions pertain to manipulating cubes.

## AddCubeDependency

AddCubeDependency lets you predefine cube inter-dependencies to avoid lock contention problems during normal system use.

In normal operations, cube dependencies are established when data which crosses cube boundaries (such as data that is derived by a rule that references an external cube) is retrieved. To create the dependency information, the server must lock the cubes while the dependency is established, potentially maintaining the lock during a long view calculation. Since this is a 'write' lock, other users are prevented from accessing the cubes. The AddCubeDependency function allows the dependency to be established when the server starts up, preventing later lock contention as no new dependency need be established.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AddCubeDependency(BaseCube, DependentCube);
```

| Argument | Description |
|---|---|
| BaseCube | The name of the cube upon which the DependentCube is dependent. |
| DependentCube | The name of a cube that depends on another cube (BaseCube) for data. Most commonly, this would be a cube that uses rules to pull data from an external cube. |

#### Example

Consider a cube named 'SalesCube' that includes the rule `['net']=!Units * DB('PriceCube', ... );`

In this example, 'SalesCube' is the dependent cube, as it is dependent on values in the base cube named 'PriceCube' to calculate the value of 'net'. To establish this dependency, you should run the following function in a TurboIntegrator process: `AddCubeDependency( 'PriceCube', 'SalesCube' );`

To establish dependency at server load time, you can create a process that runs the AddCubeDependency function, schedule the process as a chore, and then define that chore as one of the **StartupChores** in Tm1s.cfg.

## CellGetN

CellGetN retrieves a value from a numeric cube cell.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellGetN(Cube, e1, e2 [,...en]);
```

| Argument | Description |
|---|---|
| Cube | The name of the cube from which you want to retrieve a value. |
| e1,...e*n* | Dimension element names that define the intersection of the cube containing the value to be retrieved.<br><br>Arguments e1 through e*n* are sequence-sensitive. e1 must be an element from the first dimension of the cube, e2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables.<br><br>If any of the dimensions in your cube use hierarchies, you can also use the 'HierarchyName':'ElementName' convention to specify an element within a specific hierarchy. In this case, the sequence of arguments must still adhere to the order of dimensions in the cube. For example, if you want to reference multiple elements from hierarchies in the second dimension of your cube, all such arguments must appear after the argument for the first dimension in the cube and before the argument for the third dimension in the cube. |

#### Example of CELLGETN and variables

When this function is used in a conditional statement (IF), the statement is the portion containing the condition, not the entire conditional block. After a minor error, execution continues with the next statement. TurboIntegrator processing has no knowledge that it was in a conditional once the minor error is processed, so the next statement is the next line, not the line after the ENDIF;.

To avoid this situation, use variables for any operation that could encounter a minor error and then use the variables in the conditional statement.

```
V1 = CELLGETN('PNLCube', 'fred','argentina','Sales','Jan');
IF(V1 = 454);
    ASCIIOUTPUT('bug.txt', 'if logic not working properly');
ENDIF;
```

### Example of CellGetN without hierarchies

This example illustrates a function that uses simple element names as arguments and does not reference any hierarchies. It retrieves the numeric value at the intersection of the Actual, Argentina, S Series 1.8L Sedan, Sales, and Jan elements in the Sales cube.

```
CellGetN('Sales', 'Actual', 'Argentina', 'S Series1.8L Sedan', 'Sales', 'Jan');
```

### Example of CellGetN referencing multiple hierarchies

This example illustrates a function that references multiple hierarchies in the Model dimension. It retrieves the numeric value at the intersection of the Actual, Argentina, S Series (from the vehicles hierarchy in the model dimension), 2.8 Litre (from the enginesize hierarchy in the model dimension), and Jan elements in the Sales cube.

```
CellGetN('Sales', 'Actual', 'Argentina',
('vehicles':'S Series', 'enginesize':'2.8 Litre'), 'Sales', 'Jan');
```

**Note:** This example artificially breaks the line of code for easier reading.

## CellGetS

CellGetS retrieves a value from a string cube cell.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellGetS(Cube, e1, e2 [,...en]);
```

| Argument | Description |
|---|---|
| Cube | The name of the cube from which you want to retrieve a value. |
| e1,...e*n* | Dimension element names that define the intersection of the cube containing the value to be retrieved.<br>Arguments e1 through e*n* are sequence-sensitive. e1 must be an element from the first dimension of the cube, e2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables. |

See the note at "CellGetN" on page 1100 concerning IF logic with this function.

### Example

```
CellGetS('Personnel', 'Rep', 'Europe', 'Product');
```

This example retrieves the string value at the intersection of the Rep, Europe, and Product elements in the Personnel cube.

## CellIncrementN

CellIncrementN increments an existing numeric cell value by a specified value.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellIncrementN(x, Cube, e1, e2 [,...en]);
```

| Argument | Description |
|---|---|
| x | A numeric value that you want to add to an existing cell value. |
| Cube | The name of the cube to which you want to send the value. |
| e1,...e*n* | Dimension element names that define the intersection of the cube to receive the value.<br>Arguments e1 through e*n* are sequence-sensitive. e1 must be an element from the first dimension of the cube, e2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables. |

#### Example

```
CellIncrementN(1000, 'y2ksales', 'Actual', 'Argentina', 'S Series 1.8L Sedan', 'Sales', 'Jan');
```

This example increments the value at the intersection of the Actual, Argentina, S Series 1.8L Sedan, Sales, and Jan elements in the y2ksales cube by 1000.

## CellIsUpdateable

CellIsUpdateable determines whether a cube cell can be written to. The function returns 1 if the cell can be written to, otherwise it returns 0.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellIsUpdateable(Cube, e1, e2 [,...en]);
```

| Argument | Description |
|---|---|
| Cube | The name of the cube to which you want to write a value. |
| e1,...e*n* | Dimension element names that define the cell to which you want to write a value.<br>Arguments e1 through e*n* are sequence-sensitive. e1 must be an element from the first dimension of the cube, e2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables. |

**Example**

```
CellIsUpdateable ('y2ksales', 'Actual', 'Argentina','S Series 1.8L Sedan', 'Sales', 'Jan');
```

This example determines if the cell defined by the elements Actual, Argentina, S Series 1.8L Sedan, Sales, and Jan in the y2ksales cube can be written to. If the cell can receive a value, the function returns 1, otherwise it returns 0.

## CellPutN

CellPutN sends a numeric value to a cube cell.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellPutN(x, Cube, e1, e2 [,..., en]);
```

*Table 17. CellPutN arguments*

| Argument | Description |
|---|---|
| x | A numeric value. |
| Cube | The name of the cube to which you want to send the value. |
| e1, e2, ..., en | Dimension element names that define the intersection of the cube to receive the value.<br>Arguments e1 through en are sequence-sensitive. e1 must be an element from the first dimension of the cube, e2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables. |

**Note:** If you supply invalid arguments to the `CellPutN()` function in a TurboIntegrator process when the cube does not exist, an error is sent to the `tm1server.log`.

**Example**

```
CellPutN(12345, 'y2ksales', 'Actual', 'Argentina', 'S Series 1.8L Sedan', 'Sales', 'Jan');
```

This example sends the value 12345 to the intersection of the Actual, Argentina, S Series 1.8L Sedan, Sales, and Jan elements in the y2ksales cube.

## CellPutProportionalSpread

CellPutProportionalSpread distributes a specified value to the leaves of a consolidation proportional to existing cell values. CellPutProportionalSpread replaces existing cell values; it cannot be used to add to or subtract from existing cell values.

The function is analogous to the Proportional Spread data spreading method. If you must add to or subtract from existing cell values, use the Proportional Spread method, which can be executed through the user interface or through data spreading syntax.

**Note:** When using CellPutProportionalSpread to distribute a value to the leaves of a consolidation, only those leaves already containing non-zero values are changed. This is because zero values cannot be incremented or decremented proportionally; any proportion of zero is still zero.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellPutProportionalSpread( value, cube, e1, e2, e3...,en );
```

| Argument | Description |
|---|---|
| value | The value you want to distribute. |
| cube | The name of the cube into which you want to distribute the value. |
| e1...*en* | The names of the elements that identify the consolidation whose leaves will accept the distributed value.<br>Arguments e1 through e*n* are sequence-sensitive. e1 must be an element from the first dimension of the cube, e2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables. |

#### Example

```
CellPutProportionalSpread(7000,'SalesCube', 'Actual','North America',
'S Series 1.8L Sedan', 'Sales', 'Jan')
```

This example distributes the value 7000 to the children of the consolidation in the SalesCube identified by the elements Actual, North America, S Series 1.8L Sedan, Sales, and Jan.

## CellPutS

CellPutS sends a string value to a cube cell.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellPutS(String, Cube, e1, e2 [,...en]);
```

| Argument | Description |
|---|---|
| String | A string. |
| Cube | The name of the cube to which you want to send the string. |

| Argument | Description |
|---|---|
| e1,...e*n* | Dimension element names that define the intersection of the cube to receive the string.<br><br>Arguments e1 through e*n* are sequence-sensitive. e1 must be an element from the first dimension of the cube, e2 must be an element from the second dimension, and so on. These arguments can also be the names of aliases for dimension elements or TurboIntegrator variables. |

#### Example

```
CellPutS('jones', 'Personnel', 'Rep', 'Europe', 'Product');
```

This example sends the string 'jones' to the intersection of the Rep, Europe, and Product elements in the personnel cube.

## CubeClearData

CubeClearData clears all of the data in a cube. This function is much faster than doing an operation such as creating a view to cover the entire cube, and then doing a ViewZeroOut() to zero out the entire cube.

When you use CubeClearData to clear data from a cube, any cells in the cube that are fed with feeders are also cleared. You must resave the rule that establishes the feeders or use the CubeProcessFeeders function to restore the fed cells.

This function deletes only the cube data, it does not delete and re-create the cube itself. This has implications when sandboxes are used. If a cube is deleted and then re-created, any sandboxes a user may have will be discarded, since the cube against which those sandboxes were created was deleted (even though a cube may have been re-created with the same name). If, however, CubeClearData is used, the sandbox data will still be considered valid, since the cube against which the sandbox was created continues to exist.

CubeClearData is valid in processes only.

**Note:** The effect of the CubeClearData function is not recorded in the transaction log; the log will not contain any entries relating to the removal of data from the cube resulting from the use of CubeClearData.

### Syntax

```
CubeClearData( name-of-cube-as-string );
```

### Argument

The name of the cube to clear, as a string.

#### Example

```
CubeClearData( 'expense' );
```

## CubeCreate

CubeCreate creates a cube from specified dimensions. The order of dimensions specified in the function will be the order of dimensions in the cube definition. After execution, CubeCreate automatically saves the resulting .cub file to disk.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** When you create a cube using the REST API, transaction logging is enabled on the new cube. The Planning Analytics Workspace modeling workbench uses the REST API for all interactions with TM1. However, if you create a cube with the CubeCreate TurboIntegrator function, transaction logging is not enabled. To enable transaction logging, use the CubeSetLogChanges TurboIntegrator function.

### Syntax

```
CubeCreate(Cube, d1, d2 [,...dn]);
```

| Argument | Description |
|---|---|
| Cube | The name you want to assign to the cube. |
| d1,...d*n* | The names of dimensions that comprise the cube. You must specify at least two, but no more than 256, dimensions. |

#### Example

```
CubeCreate('y2ksales', 'Actvsbud', 'Region', 'Model','Account1', 'Month');
```

This example creates a cube named y2ksales using the dimensions Actvsbud, Region, Model, Account1, and Month.

## CubeDestroy

CubeDestroy deletes a specified TM1 cube.

This function is valid in TM1 TurboIntegrator processes only.

You can use CubeDestroy to delete control cubes.

### Syntax

```
CubeDestroy(Cube);
```

| Argument | Description |
|---|---|
| Cube | The name of the cube you want to delete. |

#### Example

```
CubeDestroy('y2ksales');
```

This example deletes the cube named y2ksales.

## CubeDimensionCountGet

CubeDimensionCountGet returns the number of dimensions in a cube.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CubeDimensionCountGet(CubeName);
```

| Argument | Description |
|---|---|
| CubeName | The name of the cube for which you want to determine the number of dimensions. |

### Example

```
CubeDimensionCountGet('Sales');
```

In this example, the function returns the number of dimensions in the Sales cube.

## CubeExists

CubeExists determines whether a specific cube exists on the server from which a TurboIntegrator process is executed. The function returns 1 if the cube exists on the server, otherwise it returns 0.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CubeExists(CubeName);
```

| Argument | Description |
|---|---|
| CubeName | The name of the cube whose existence you want to confirm. |

#### Example

```
CubeExists('Inventory');
```

This example determines if the Inventory cube exists on the server.

## CubeGetLogChanges

CubeGetLogChanges returns the Boolean value of the Logging property for a specified cube.

The Logging property is set in the Security Assignments dialog box and stored in the }CubeProperties control cube. If Logging is turned on for a cube, the function returns 1. If logging is turned off the function returns 0.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
CubeGetLogChanges(CubeName);
```

| Argument | Description |
|---|---|
| CubeName | The cube for which you want to return the value of the Logging property. |

#### Example

```
CubeGetLogChanges('2002sales');
```

If Logging is turned on for the 2002sales cube, the function returns 1.

## CubeSaveData

CubeSaveData() serializes a cube.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

To improve performance, transaction logging may be disabled while loading data. To safeguard newly loaded data in the unlikely event of a server crash, the changes can be serialized to disk. SaveDataAll has been used to serialize data to disk and to truncate the transaction log. When processing a SaveDataAll command, the server acquires a READ lock on every cube and an IX lock on every changed cube. This can cause significant contention with user activity if SaveDataAll is run during periods of user activity.

Typically not all the cubes affected by SaveDataAll need to be serialized since not all cubes are typically loaded with new data. CubeSaveData is used to serialize an individual cube to disk. CubeSaveData serializes the cube's data that has been committed to memory including the modifications that have been performed against it in the current TurboIntegrator process but not yet committed.

### Syntax

```
CubeSaveData(Cube);
```

| Argument | Description |
|---|---|
| Cube | The name of the cube you want to serialize. |

#### Example

```
CubeSaveData ('SalesCube');
```

Consider the following TurboIntegrator process code:

```
CellPutN(500, 'y2ksales', 'Actual', 'Argentina', 'S Series 1.8 L Wagon', 'Sales', 'Jan');
CubeSaveData('y2ksales');
CellPutN(1000, 'y2ksales', 'Actual', 'Argentina', 'S Series 1.8 L Wagon', 'Sales', 'Jan');
```

When the CubeSaveData command is processed, the value of 500 for the January Sales cell will be included in the cube's serialization to disk, even though it has not yet been committed. The update of the January Sales cell to 1000 will not be part of the serialization.

### Transaction Log

A new transaction entry appears in the Transaction log when CubeSaveData has been run. When processing a transaction log file during recovery, all updates to a cube that have been applied so far will be discarded when a CubeSaveData directive against the cube is encountered as all of the updates have already been serialized to the cube.

### Server Crash Recovery

The SaveDataAll command takes advantage of the fact that all cubes are locked during its processing and truncates the transaction log knowing that all updates performed before serialization have been safely stored to disk. This is not the case for CubeSaveData so you must modify the way data recovery is performed when a cube has been serialized.

The transaction log file could contain records that represent changes that are older than the most recent data in the cube and should not be applied when data is being recovered.

## CubeSetConnParams

CubeSetConnParams is used to encrypt the password for a virtual cube in the }CubeProperties cube.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
CubeSetConnParams(cubeName, providerName, dataSourceLocation,dataSourceName,
dataSourceCatalog, userID, password, sapClientID, sapClientLang, providerString);
```

| Argument | Description |
|---|---|
| cubeName | The name of the cube for which you want to set the password. |
| providerName | |
| dataSourceLocation | Name your administrator assigns to a set of catalogs at a particular location. In Microsoft Analysis Services, this is the name of a registered server. |
| dataSourceName | |
| dataSource catalog | The name assigned by your administrator to a particular collection of databases (Cubes, Dimensions and other objects). For MAS, this is the name of the database. |
| UserID | A valid username for the database. |
| Password | Password to use for this data source. |
| sapClientID | SAP client ID |
| sapClientLang | SAP language setting. |
| providerString | |

#### Example

```
CubeSetConnParams(sc, TM1OLAP, tm1server, , sdata, admin, apple, , , );
```

## CubeSetLogChanges

CubeSetLogChanges sets the Logging property for a cube.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
CubeSetLogChanges(Cube, LogChanges);
```

| Argument | Description |
|---|---|
| Cube | The name of the cube for which you want to set the LOGGING property. |
| LogChanges | The Boolean value you want to assign to the property. 1= LOGGING on, 0 = LOGGING off. |

## CubeTimeLastUpdated

CubeTimeLastUpdated returns a serial value that indicates the date and time at which a specified cube was last updated.

The serial value that is returned by this function uses a starting time of Jan 1 1900 12:00:00 A.M., which is equivalent to the value 1.0. Dates are represented by integers, while times are represented as decimal numbers between .0 and .999999. This is consistent with the way date and time serial values are stored and reported in Microsoft Excel.

**Note:** By default, TM1 date and time serial values use a starting time of Jan 1 1960 12:00:00 A.M. To resolve the inconsistency between Excel and TM1 date and time serial values, you can set **`UseExcelSerialDate=T`** in your Tm1s.cfg file to instruct the TM1 server to use date and time serial values that conform to Excel standards.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CubeTimeLastUpdated(cube);
```

| Argument | Description |
|---|---|
| cube | The name of the cube. |

#### Example

```
CubeTimeLastUpdated('Sales');
```

This example returns a value corresponding to the time when the Sales cube was last updated.

## CubeUnload

CubeUnload unloads a specified cube, along with all associated cube views, from memory.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CubeUnload(CubeName);
```

| Argument | Description |
|---|---|
| CubeName | The cube you want to unload from memory. |

**Example**

```
CubeUnload('ManufacturingBudget');
```

This example unloads the ManufacturingBudget cube, and any associated views, from server memory.

# Data Reservation TurboIntegrator Functions

Use the following process functions to programmatically obtain, release and manage Data Reservations.

Data Reservation functions are not valid in processes on TM1 Database 12.

For more details about using the Data Reservation feature, see "Using Data Reservations" in the IBM Cognos *TM1 for Developers* documentation.

## CubeDataReservationAcquire

CubeDataReservationAcquire acquires a Data Reservation for the specified cube, user and tuple.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
CubeDataReservationAcquire(Cube, User, bForce, Address, [AddressDelimiter])
```

| Argument | Description |
|---|---|
| Cube | Name of the cube. |
| User | Name of the owner for the new reservation.<br>The user name supplied will be validated to make sure it is an existing user. |
| bForce | Boolean value that determines the behavior if the requested reservation conflicts with an existing reservation.<br>If set to 0 (false), then the request is rejected if it conflicts with an existing reservation.<br>If set to 1 (true) and the user running the TurboIntegrator process has the DataReservationOverride capability, then the conflicting reservations are released, and the requested one is granted. |
| Address | Tokenized string sequence of element names that define the tuple. The order must match the original dimension order of the cube.<br>All the cells in the cube contained by the tuple make up the region being reserved. You can choose one element from each dimension or use an empty string between the delimiters to select an entire dimension. Depending on where the element is located in the hierarchy, the request reserves a single cell, a slice, or the entire cube. |
| AddressDelimiter | Optional character string that is used to separate element names in the Address parameter.<br>Default value is '\|'. |

### Return Value

Boolean - returns true if the acquisition succeeded.

#### Example

```
CubeDataReservationAcquire('DRTestCube','User1',0,'ElemX|ElemY|ElemZ');
```

The following example sets the bForce parameter to 1 to force the DR request if a conflict exists and uses a different delimiter character for the AddressDelimiter parameter.

```
CubeDataReservationAcquire('DRTestCube','User2',1,'ElemX*ElemY*ElemZ','*');
```

## CubeDataReservationRelease

CubeDataReservationRelease releases the specified Data Reservation.

If the user specified is not the same as the owner of the reservation, then the release will only succeed if the user specified has the DataReservationOverride capability enabled.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
CubeDataReservationRelease(Cube, User, Address,[AddressDelimiter])
```

| Argument | Description |
|---|---|
| Cube | Name of the cube. |
| User | Name of the owner of the reservation.<br>The user name supplied will be validated to make sure it is an existing user. |
| Address | Tokenized string sequence of element names that define the tuple. The order must match the original dimension order of the cube. |
| AddressDelimiter | Optional character string that is used to separate element names in the Address parameter.<br>Default value is '\|'. |

### Return Value

Boolean - returns true if the release succeeded.

#### Example

```
CubeDataReservationRelease('DRTestCube','User1','ElemX|ElemY|ElemZ');
```

The following example uses a different character for the AddressDelimiter parameter.

```
CubeDataReservationRelease('DRTestCube','User2','ElemX*ElemY*ElemZ','*');
```

## CubeDataReservationReleaseAll

CubeDataReservationReleaseAll releases multiple existing Data Reservations.

All reservations fully contained by the specified address that match the user filter will be released. A blank user filter means all users.

If the user filter specified is not the same as the user running the TurboIntegrator proces, then the DataReservationOverride capability must be enabled.

Using a blank user filter and all wildcards in the address field releases all reservations.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
CubeDataReservationReleaseAll(Cube, UserFilter, Address, [AddressDelimiter])
```

| Argument | Description |
|---|---|
| Cube | Name of the cube. |
| UserFilter | User name filter to match against existing reservations. |
| Address | Tokenized string sequence of element names that define the tuple. The order must match the original dimension order of the cube. |
| AddressDelimiter | Optional character string that is used to separate element names in the Address parameter.<br>Default value is '\|'. |

### Return Value

Boolean - returns true if no errors.

#### Example

```
CubeDataReservationReleaseAll('DRTestCube','User1','ElemX|ElemY|ElemZ');
```

The following example releases all reservations in the specified cube for all users.

```
CubeDataReservationReleaseAll('DRTestCube','','||');
```

## CubeDataReservationGet

CubeDataReservationGet finds existing reservations on a specific cube for all or one user.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
CubeDataReservationGet(Index, Cube, User, [AddressDelimiter]) returns Address;
```

| Argument | Description |
|---|---|
| Index | A one-based loop index to use for iterating through reservations on the specified cube. |
| Cube | Name of the cube to search. |
| User | Reservation owner name to use as a filter.<br>If left blank, the function returns reservations for any owner.<br>If a name is provided, the function filters the results for just the specified owner. |
| AddressDelimiter | Optional character string that is used to separate element names in the returned Address parameter.<br>Default value is '\|'. |

## Return Value

Address - Reservation creation time, name of the reservation owner and Element address of the reservation. Creation time comes first, followed by delimiter, followed by UserID, followed by delimiter, followed by Elements IDs separated by the delimiter in order of dimensions in the cube (original order).

An empty string is returned if there is no entry for the specified index.

The format of the return value is:

`[creation time][delimiter][owner name][delimiter][element1][delimiter][element2][delimiter]…[elementN]`

For example:

"20100622211601|Fred Bloggs|Element1|Element2|Element3"

**Note:** The reservations can change while iterating the list of reservations so the use of index is not guaranteed to give a complete list of reservations. Reservations can be added or removed at any position in the list, so reservations can be skipped or repeated when looping through index values.

If the owner filter is specified, then the index applies only to the members of the filtered list. If the list of reservations has owners as follows: User1, User1, User2 and the request specifies an owner of User2 then an index of 1 will retrieve the third member of the list.

### Example

```
CubeDataReservationGet(1,'DRTestCube','User1','*');
```

```
CubeDataReservationGet(1,'DRTestCube','');
```

The following sample would find all the reservations owned by user Fred Bloggs in the Expense Input cube and do "something useful" with them:

```
vIndex = 1;
vCube = 'Expense Input';
vUserFilter = 'Fred Bloggs';
vDelim = '|';
vAddress = CubeDataReservationGet( vIndex, vCube, vUserFilter,vDelim);
WHILE (vAddress @<> '');
    vSep1 = SCAN( vDelim, vAddress);
        vDRUser = SUBST( vAddress, 1, vSep1 - 1);
        vDRAddress = SUBST( vAddress, vSep1 + 1, LONG(vDRAddress) - vSep1);

#     do something meaningful with the
user and reservation address here
```

```
        vIndex = vIndex + 1;
        vAddress = CubeDataReservationGet( vIndex, vCube, vUserFilter,vDelim);
END;
```

## CubeDataReservationGetConflicts

CubeDataReservationGetConflicts finds existing reservations on a specific cube that would conflict with the specified user, address and tuple.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
CubeDataReservationGetConflicts(Index, Cube, User, Address, [AddressDelimiter])returns
ConflictAddress;
```

| Argument | Description |
|---|---|
| Index | A one-based loop index to use for iterating through conflicts that satisfy this query. |
| Cube | Name of the cube to search |
| User | The query will search for reservations that will conflict with this user. |
| Address | Tokenized string sequence of element names that define the tuple. The order must match the original dimension order of the cube. |
| AddressDelimiter | Optional character string that is used to separate element names in the Address parameter.<br>Default value '\|'. |

### Return Value

ConflictAddress - Reservation creation time, name of the reservation owner and Element address of the reservation. The creation time comes first, followed by delimiter, followed by UserID, followed by delimiter, followed by Elements IDs separated by the delimiter in order of dimensions in the cube (original order).

An empty string is returned if there is no entry for the specified index.

The format of the return value is:

```
                [creation time][delimiter][owner name][delimiter][element1][delimiter]
[element2][delimiter]…[elementN]
```

For example:

"20100622211601|Fred Bloggs|Element1|Element2|Element3"

**Note:** The reservations can change while iterating the list of conflict reservations so the use of index is not guaranteed to give a complete list of reservations. Reservations can be added or removed at any position in the list, so reservations can be skipped or repeated when looping through index values.

# Date and Time TurboIntegrator Functions

These functions format and parse dates and times in a wide variety of formats and locales.

## FormatDate

FormatDate formats a date value according to a formatter defined with the NewDateFormatter function.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
FormatDate(Date, <Pattern>, <Index>)
```

| Argument | Description |
|---|---|
| *Date* | A date value.<br>The type of value, serial or UNIX, should correspond to the formatter used. |
| *Pattern* | Pattern used for formatting dates.<br>Refer to `https://unicode-org.github.io/icu/userguide/format_parse/datetime` for a complete list of format syntax.<br>If an empty string is passed, then the format is determined by the locale based on the FormatterStyle and FormatterType parameters that were used with the NewDateFormatter function. |
| *Index* | Index returned by a call to the NewDateFormatter function.<br>The default value is 0.<br>If no date formatter exists at the index, then a default formatter is used as though it had been created with the following call:<br>`NewDateFormatter('', 'Etc/UTC', 'serial', 'medium', 'date')` |

### Example

```
sDate = FormatDate(18000);
```

## NewDateFormatter

NewDateFormatter defines a date formatter. It returns an index for use in the ParseDate and FormatDate functions. The indices start at 0 and go up by one for each call to NewDateFormat. Date formatters are valid during execution of the process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
NewDateFormatter(Locale, <TimeZone>, <UseUNIXTime>, <FormatterStyle>, <FormatterType>,
<TimeType>)
```

| Argument | Description |
|---|---|
| *Locale* | Locale used for parsing or formatting dates.<br>If an empty string is passed, then the operating system locale is used. Locales are specified in the format language[_territory][.variant]. For example, `cs_CK` is the Czech language and Czech Republic. |

| Argument | Description |
|---|---|
| *TimeZone* | Timezone used for parsing or formatting dates.<br>Refer to http://en.wikipedia.org/wiki/List_of_tz_database_time_zones for a complete list of time zones.<br>If not specified, the time zone used is `UTC ('Etc/UTC')`. |
| *UseUNIXTime* | If `'unix'` is specified, then times are treated as milliseconds since January 1, 1970. Otherwise, they are treated in TM1 serial format.<br>Note that only dates later than January 1, 1970 can be processed even if TM1 serial format is used. |
| *FormatterStyle* | Controls the date format used when an empty pattern is specified to the FormatDate or ParseDate functions.<br>Valid values are `'full'`, `'long'`, `'medium'` or `'short'`.<br>The default is `'medium'`. |
| *FormatterType* | Controls the type of format used when an empty pattern is specified to the FormatDate or ParseDate functions.<br>Valid values are `'time'`, `'date'` or `'datetime'`.<br>The default is `'date'`. |

### Example

```
dfUNIX = NewDateFormatter('', 'Etc/UTC', 'unix');
```

```
dfStyleFullDateTime = NewDateFormatter('en_us', 'America/Toronto', 'serial', 'full',
'datetime');
```

## ParseDate

ParseDate parses a date string according to a formatter defined with the NewDateFormatter function.

A date value that is either serial or UNIX, depending on the formatter specified, is returned. If the date cannot be parsed then an undefined value is returned. This can be tested with the ISUND function.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ParseDate (DateString, <Pattern>, <Index>)
```

| Argument | Description |
|---|---|
| *DateString* | A date string. |

| Argument | Description |
|---|---|
| *Pattern* | Pattern used for parsing dates.<br>Refer to `https://unicode-org.github.io/icu/userguide/format_parse/datetime` for a complete list of format syntax.<br>If an empty string is passed, then the format is determined by the locale based on the FormatterStyle and FormatterType parameters that were used with the NewDateFormatter function. |
| *Index* | Index returned by a call to the NewDateFormatter function. The default value is 0. If no date formatter exists at the index, then a default formatter is used as though it had been created with the following call:<br>`NewDateFormatter('', 'Etc/UTC', 'serial', 'medium', 'date')` |

### Example

```
nDate = ParseDate('2011/11/24', 'yyyy/MM/dd');
```

## Dimension Manipulation TurboIntegrator Functions

These functions facilitate the manipulation of dimensions.

### DimensionCreate

DimensionCreate creates a new dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionCreate(DimName);
```

| Argument | Description |
|---|---|
| DimName | The name you want to assign to the dimension. |

#### Example

```
DimensionCreate('Product');
```

This example creates the Product dimension.

### DimensionDeleteAllElements

DimensionDeleteAllElements deletes all the elements in a dimension. This function is useful for recreating dimension hierarchies.

**Note:** Deleting an element deletes all cube data identified by that element. However, if you use DimensionDeleteAllElements to delete elements, then recreate those elements with the same names in the Metadata tab, any data points in a cube identified by the elements will be retained after rebuilding the dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionDeleteAllElements(DimName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension from which you want to delete all elements. |

#### Example

```
DimensionDeleteAllElements('Model');
```

This example deletes all elements in the Model dimension.

## DimensionDeleteElements

DimensionDeleteElements deletes all elements from a dimension using the subset of elements. All elements in the referenced subset are deleted, including C level elements.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionDeleteElements (DimensionName, Subset )
```

| Argument | Description |
|---|---|
| DimensionName | The name of the dimension from which you want to delete the subset of elements. |
| Subset | The list of elements to delete from the indicated dimension. The subset is usually temporary. |

## DimensionDestroy

DimensionDestroy deletes a dimension from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionDestroy(DimName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension you want to delete. |

#### Example

```
DimensionDestroy('Product');
```

This example deletes the Product dimension from the TM1 database.

## DimensionElementComponentAdd

DimensionElementComponentAdd adds a component (child) to a consolidated element. You can't use this function in the Epilog procedure of a TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionElementComponentAdd(DimName, ConsolidatedElName,ElName, ElWeight);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the consolidated element to which you want to add a child. |
| ConsolidatedElName | The element to which you want to add a child. |
| ElName | The name of the child element. |
| ElWeight | The weight of the child element. |

#### Example

```
DimensionElementComponentAdd('Measures', 'Net Sales', 'Expenses', -1);
```

This example adds the child Expenses to the Net Sales consolidation in the Measures dimension. The child has a weight of -1 in the consolidation.

## DimensionElementComponentAddDirect

DimensionElementComponentAddDirect adds a component (child) to a consolidated element by directly editing a dimension.

This function is valid in TM1 TurboIntegrator processes only.

The default means of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like DimensionElementComponentAdd) are used in the Metadata procedure of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element modification needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
DimensionElementComponentAddDirect(DimName, ConsolidatedElName,ElName, ElWeight);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the consolidated element to which you want to add a child. |
| ConsolidatedElName | The consolidated element to which you want to add a child. |
| ElName | The name of the child element. |
| ElWeight | The weight of the child element. |

#### Example

```
DimensionElementComponentAddDirect('Measures', 'Net Sales', 'Expenses', -1);
```

This example adds the child Expenses to the Net Sales consolidation in the Measures dimension. The child has a weight of -1 in the consolidation.

## DimensionElementComponentDelete

DimensionElementComponentDelete deletes a component (child) from a consolidated element.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionElementComponentDelete(DimName, ConsolidatedElName,ElName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the consolidated element from which you want to delete a child. |
| ConsolidatedElName | The consolidated element from which you want to delete a child. |
| ElName | The name of the child element you want to delete. |

#### Example

```
DimensionElementComponentDelete('Region', 'Benelux','Belgium');
```

This example deletes the Belgium child from the Benelux consolidation in the Region dimension.

## DimensionElementComponentDeleteDirect

DimensionElementComponentDeleteDirect deletes a component (child) from a consolidated element by directly editing the dimension.

This function is valid in TM1 TurboIntegrator processes only.

The default means of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like DimensionElementComponentDelete) are used in the Metadata procedure of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element modification needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
DimensionElementComponentDeleteDirect(DimName, ConsolidatedElName,ElName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the consolidated element from which you want to delete a child. |
| ConsolidatedElName | The consolidated element from which you want to delete a child. |
| ElName | The name of the child element you want to delete. |

#### Example

```
DimensionElementComponentDeleteDirect('Region', 'Benelux','Belgium');
```

This example deletes the Belgium child from the Benelux consolidation in the Region dimension.

## DimensionElementDelete

DimensionElementDelete deletes an element from a dimension.

**Note:** Deleting an element deletes all cube data identified by that element.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionElementDelete(DimName, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension that contains the element you want to delete. |

| Argument | Description |
|---|---|
| ElName | The element you want to delete. |

**Example**

```
DimensionElementDelete('Region', 'Belgium');
```

This example deletes the element Belgium from the Region dimension.

## DimensionElementDeleteDirect

DimensionElementDeleteDirect deletes an element from a dimension by directly editing the dimension.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** Deleting an element deletes all cube data identified by that element.

The default means of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like DimensionElementDelete) are used in the Metadata procedure of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element modification needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
DimensionElementDeleteDirect(DimName, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension that contains the element you want to delete. |
| ElName | The element you want to delete. |

**Example**

```
DimensionElementDeleteDirect('Region', 'Belgium');
```

This example deletes the element Belgium from the Region dimension.

## DimensionElementExists

DimensionElementExists determines whether a specific element exists in a dimension on the server from which a TurboIntegrator process is executed. The function returns 1 if the element exists in the dimension on the server, otherwise it returns 0.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionElementExists(DimName, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension that contains the element that you want to find. The dimension must exist on the server where the TurboIntegrator process is executed. |
| ElName | The element that you want to find. The ElName argument accepts both the element name and the alias. |

#### Example

This example determines whether the element Belgium exists in the Region dimension on the server.

```
DimensionElementExists('Region', 'Belgium');
```

## DimensionElementInsert

DimensionElementInsert adds an element to a dimension. You can use this function to add numeric, string, or consolidated elements. You can't use this function in the Data or Epilog procedures of a TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionElementInsert(DimName, InsertionPoint, ElName,ElType);
```

| Argument | Description |
|---|---|
| DimName | The dimension to which you want to add an element. |
| InsertionPoint | An existing dimension element. The element being added to the dimension will be inserted immediately before this existing element. If this parameter is an empty string (''), the new element is added to the end of the dimension. |
| ElName | The name that you want to assign to the new element. |

| Argument | Description |
|---|---|
| ElType | The element type. There are three possible ElType values:<br>N - Signifies a numeric element.<br>S - Signifies a string element.<br>C - Signifies a consolidated element. |

#### Example

```
DimensionElementInsert('Region','Belgium','Netherlands','S');
```

This example adds the string element Netherlands to the Region dimension. Netherlands is added immediately before Belgium in the dimension.

#### Example with an empty insertion point

```
DimensionElementInsert('Region','','Netherlands','S');
```

This example adds the string element Netherlands to the Region dimension. Netherlands is added to the end of the dimension.

## DimensionElementInsertDirect

DimensionElementInsertDirect adds an element to a dimension by directly editing the dimension. You can use this function to add numeric, string, or consolidated elements.

This function is valid in TM1 TurboIntegrator processes only.

The default method of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like DimensionElementInsert) are used in the metadata tab of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element insertion needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

## Syntax

```
DimensionElementInsertDirect(DimName, InsertionPoint, ElName,ElType);
```

| Argument | Description |
|---|---|
| DimName | The dimension to which you want to add a new element. |
| InsertionPoint | An existing dimension element. The element being added to the dimension will be inserted immediately before this existing element. If this parameter is empty, the new element is added to the end of the dimension.<br>Note that this function is optimized for the case where the InsertionPoint is passed as an empty string. |
| ElName | The name you want to assign to the new element. |
| ElType | The element type. There are three possible ElType values:<br>N - Signifies a numeric element.<br>S - Signifies a string element.<br>C - Signifies a consolidated element. |

**Example**

```
DimensionElementInsertDirect('Region', 'Belgium', 'Netherlands','N');
```

This example adds the numeric element Netherlands to the Region dimension. Netherlands displays immediately before Belgium in the dimension definition.

## DimensionElementPrincipalName

DimensionElementPrincipalName returns the principal name of an element or element alias.

TurboIntegrator must use principal element names when updating dimensions; element aliases cannot be used. This function is useful for determining principal element names while attempting to update a dimension when only element aliases are available to the TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

## Syntax

```
DimensionElementPrincipalName( DimName, ElName )
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension from which you want to retrieve a principal element name. |
| ElName | An element name. ElName can be either an element alias or a principal element name. |

**Example**

If ElName is not in the currently saved version of DimName, the function returns ElName.

If ElName is in DimName, whether as an element alias or a principal element name, it returns the principal name of the element.

## DimensionExists

DimensionExists determines whether a specific dimension exists on the server from which a TurboIntegrator process is executed. The function returns 1 if the dimension exists on the server, otherwise it returns 0.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionExists(DimName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension whose existence you want to confirm. |

#### Example

```
DimensionExists('Region');
```

This example determines if the Region dimension exists on the server.

## DimensionHierarchyCreate

DimensionHierarchyCreate creates a new hierarchy in an existing dimension. The hierarchy cannot have the same name as the dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionHierarchyCreate(DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | The name of the existing dimension that will contain the hierarchy. |
| HierName | The name that you want to assign to the hierarchy. You cannot use the name of the dimension. |

#### Example

```
DimensionHierarchyCreate('Vehicles', 'Trucks');
```

This example creates the empty Trucks hierarchy in the Vehicles dimension.

## DimensionSortOrder

DimensionSortOrder sets a sort type and sense for dimension elements and for components of consolidated elements within a dimension. The sort order defined by DimensionSortOrder determines how the subset displays in the Subset Editor.

DimensionSortOrder sets properties for a dimension; the dimension is not actually sorted until it is saved on the server.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionSortOrder(DimName, CompSortType, CompSortSense, ElSortType , ElSortSense);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension for which you want to set a sort order. |
| CompSortType | Defines how components of consolidated elements appear in the dimension. There are two CompSortType values:<br>ByInput - Retains the order in which components were originally inserted into consolidations.<br>ByName - Sorts components of consolidations by name. |
| CompSortSense | Defines the sort sense for components of consolidations. This is a required argument, but it applies only when the CompSortType is ByName. There are two possible CompSortSense values:<br>Ascending - Sorts components of consolidations in ascending alphabetical order.<br>Descending - Sorts components of consolidations in descending alphabetical order. |
| ElSortType | Defines a sort order for dimension elements. There are four possible ElSortType values:<br>ByInput - Retains the order in which elements were originally inserted into the dimension.<br>ByName - Sorts dimension elements by name.<br>ByLevel - Sorts dimension elements by level.<br>ByHierarchy - Sorts dimension elements by hierarchy. |

| Argument | Description |
|---|---|
| ElSortSense | Defines the sort sense for dimension elements. This is a required argument, but it applies only when the ElSortType is ByName or ByLevel. There are two possible ElSortSense values:<br>Ascending - Sorts dimension elements in ascending order, either alphabetically or by level.<br>Descending - Sorts dimension elements in descending order, either alphabetically or by level. |

**Example**

```
DimensionSortOrder ('Region', 'ByName', 'Descending','ByLevel', 'Ascending');
```

This example sets a sort order for the Region dimension. All dimension elements are sorted by level in ascending order, and any components of consolidations are sorted in descending alphabetical order.

## DimensionTimeLastUpdated

DimensionTimeLastUpdated returns a serial value that indicates the date and time at which a specified dimension was last updated.

The serial value returned by this function uses a starting time of Jan 1 1900 12:00:00 A.M., which is equivalent to the value 1.0. Dates are represented by integers, while times are represented as decimal numbers between .0 and .999999. This is consistent with the way date/time serial values are stored and reported in Microsoft Excel.

**Note:** By default, TM1 date/time serial values use a starting time of Jan 1 1960 12:00:00 A.M. To resolve the inconsistency between Excel and TM1 date/time serial values, you can set `UseExcelSerialDate=T` in your Tm1s.cfg file to instruct the TM1 server to use date/time serial values that conform to Excel standards.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionTimeLastUpdated(dimension);
```

| Argument | Description |
|---|---|
| dimension | The name of the dimension. |

**Example**

```
DimensionTimeLastUpdated('Region');
```

This example returns information on when the Region dimension was last updated.

## DimensionTopElementInsert

DimensionTopElementInsert creates a root element in a dimension. If the dimension already has a single root, then this element will not be created.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DimensionTopElementInsert(DimName, InsertionPoint, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension for which you want to create a root element. |
| InsertionPoint | An existing dimension element. The root element being added to the dimension will be inserted immediately before this existing element. |
| ElName | The name you want to assign to the new root element. |

#### Example

```
DimensionTopElementInsert('Region', 'Netherlands', 'World');
```

This example adds the root element World to the Region dimension. World is inserted displays immediately before Netherlands in the dimension definition.

## DimensionTopElementInsertDirect

DimensionTopElementInsertDirect creates a root element in a dimension by directly editing the dimension. If the dimension already has a single root, then this element will not be created.

This function is valid in TM1 TurboIntegrator processes only.

The default means of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like DimensionTopElementInsert) are used in the Metadata procedure of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element modification needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
DimensionTopElementInsertDirect(DimName, InsertionPoint, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension for which you want to create a root element. |
| InsertionPoint | An existing dimension element. The root element being added to the dimension will be inserted immediately before this existing element. |
| ElName | The name you want to assign to the new root element. |

**Example**

```
DimensionTopElementInsertDirect('Region', 'Netherlands', 'World');
```

This example adds the root element World to the Region dimension. World is inserted displays immediately before Netherlands in the dimension definition.

### DimensionUpdateDirect

DimensionUpdateDirect performs a full rewrite of a dimension that has been subject to direct editing in a TurboIntegrator process, essentially compacting the memory footprint of the dimension.

A dimension that undergoes a series of direct-only edits (element deletions, in particular) will eventually use more memory than its fully-rewritten counterpart would. This function can optionally be used after directly editing a dimension with DimensionElementInsertDirect, DimensionElementDeleteDirect, DimensionElementComponentAddDirect, DimensionElementComponentDeleteDirect, and/or DimensionTopElementInsertDirect. Calling DimensionUpdateDirect incurs an initial full-copy memory cost, however it can be used to guarantee that the dimension is at its smallest possible memory footprint after processing is complete.

This function is valid in TM1 TurboIntegrator processes only.

#### Syntax

```
DimensionUpdateDirect(DimName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension you want to rewrite. |

**Example**

```
DimensionUpdateDirect('Region');
```

This example rewrites the Region dimension.

## Hierarchy Manipulation TurboIntegrator Functions

These functions facilitate hierarchy manipulation.

## CreateHierarchyByAttribute

CreateHierarchyByAttribute creates a simple 3-level hierarchy from a single attribute.

The new hierarchy consists of a single high-level root element, a middle-level of consolidations representing existing attribute values, and a lower-level of dimension leaves that include the associated attribute value.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

**Note:** This function creates a hierarchy from the current set of attribute values, but the system does not automatically keep the hierarchy in-sync with the attribute data as it changes. Modelers must regenerate the hierarchy as needed.

### Syntax

```
CreateHierarchyByAttribute(DimName, AttrName [, emptyParent [, rootName ] ] );
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension that contains the attribute. A hierarchy of the same name as the dimension will be created. |
| AttrName | The name of the attribute to create the hierarchy from. |
| emptyParent | Specifies the name of a consolidation to create, which collects dimension leaves that don't have an attribute value. If passed as an empty string, the function does not create a consolidation. |
| rootName | Overrides the root element name which by default is named after the attribute. |

#### Example

```
CreateHierarchyByAttribute ('Country', 'City');
```

This example creates a hierarchy from the City attribute in the Country dimension.

## HierarchyContainsAllLeaves

HierarchyContainsAllLeaves returns true only if the specified hierarchy contains the full set of leaf elements that are present in the dimension. That is, it contains all the leaf elements that can be seen in the special Leaves hierarchy. If the specified hierarchy is missing one or more leaf elements, this function returns false.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyContainsAllLeaves(DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension that contains the all leaves hierarchy. |
| HierName | The name of the hierarchy you are determining as an all leaves hierarchy. |

**Example**

```
HierarchyContainsAllLeaves('Region', 'Leaves');
```

This example determines if the Leaves hierarchy, in the Region dimension, contains all leaf members.

## HierarchyCreate

HierarchyCreate creates a new hierarchy in an existing dimension. The hierarchy cannot have the same name as the dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyCreate(DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | The name of the existing dimension that will contain the hierarchy. |
| HierName | The name you want to assign to the hierarchy. You cannot use the name of the dimension. |

**Example**

```
HierarchyCreate('Vehicles', 'Trucks');
```

This example creates the empty Trucks hierarchy in the Vehicles dimension.

## HierarchyDeleteAllElements

HierarchyDeleteAllElements deletes all the elements in a hierarchy. This function is useful for recreating dimension hierarchies.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyDeleteAllElements(DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension from which you want to delete all elements. |
| HierName | The name of the hierarchy within the dimension. |

**Example**

```
HierarchyDeleteAllElements('Equipment','Helmets');
```

This example deletes all elements in the Helmets hierarchy in the Equipment dimension.

## HierarchyDeleteElements

HierarchyDeleteElements deletes elements from a hierarchy using a subset of elements.

This function is valid only in TurboIntegrator processes.

### Syntax

```
HierarchyDeleteElements (DimensionName, HierarchyName, Subset)
```

| Argument | Description |
|---|---|
| DimensionName | The name of the dimension from which you want to delete the subset of elements. |
| HierarchyName | The name of the hierarchy from which you want to delete the subset of elements.<br>If the indicated hierarchy is the Leaves hierarchy, then the subset should list those leaves that should be deleted, and they are removed completely from the dimension. |
| Subset | The name of the subset that contains the elements you want to delete. The function deletes all elements in the subset from the specified hierarchy, whether leaf or consolidated. |

## HierarchyDestroy

HierarchyDestroy deletes a hierarchy from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyDestroy(DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension that contains the hierarchy. |
| HierName | The name of the hierarchy you want to delete. |

**Example**

```
HierarchyDestroy('Product','Transmissions');
```

This example deletes the Transmissions hierarchy from the TM1 database.

## HierarchyElementComponentAdd

HierarchyElementComponentAdd adds a component (child) to a consolidated element. You can't use this function in the Epilog procedure of a TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyElementComponentAdd(DimName, HierName, ConsolidatedElName, ElName, ElWeight);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the consolidated element to which you want to add a child. |
| HierName | The hierarchy of the specified dimension. |
| ConsolidatedElName | The element to which you want to add a child. |
| ElName | The name of the child element. |
| ElWeight | The weight of the child element. |

#### Example

```
HierarchyElementComponentAdd('Measures', 'Europe', 'Net Sales', 'Expenses',
-1);
```

This example adds the child Expenses to the Net Sales consolidation in the Europe hierarchy of the Measures dimension. The child has a weight of -1 in the consolidation.

## HierarchyElementComponentAddDirect

HierarchyElementComponentAddDirect adds a component (child) to a consolidated element by directly editing a dimension.

This function is valid in TM1 TurboIntegrator processes only.

The default method of editing a dimension in Cognos TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like HierarchyElementComponentAdd) are used in the Metadata procedure of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element modification needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
HierarchyElementComponentAddDirect(DimName, HierName, ConsolidatedElName, ElName, ElWeight);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the consolidated element to which you want to add a child. |
| HierName | The hierarchy of the specified dimension. |
| ConsolidatedElName | The consolidated element to which you want to add a child. |
| ElName | The name of the child element. |
| ElWeight | The weight of the child element. |

#### Example

```
HierarchyElementComponentAddDirect('Measures', 'Europe', 'Net Sales',
'Expenses', -1);
```

This example adds the child Expenses to the Net Sales consolidation in the Europe hierarchy of the Measures dimension. The child has a weight of -1 in the consolidation.

## HierarchyElementComponentDelete

HierarchyElementComponentDelete deletes a component (child) from a consolidated element.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyElementComponentDelete(DimName, HierName, ConsolidatedElName, ElName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the consolidated element from which you want to delete a child. |
| HierName | The name of the hierarchy within the dimension. |
| ConsolidatedElName | The consolidated element from which you want to delete a child. |
| ElName | The name of the child element you want to delete. |

#### Example

```
HierarchyElementComponentDelete('Region', 'Western', 'Benelux', 'Belgium');
```

This example deletes the Belgium child from the Benelux consolidation in the Western hierarchy of the Region dimension.

## HierarchyElementComponentDeleteDirect

HierarchyElementComponentDeleteDirect deletes a component (child) from a consolidated element by directly editing the dimension.

This function is valid in TM1 TurboIntegrator processes only.

The default method of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like HierarchyElementComponentDelete) are used in the Metadata procedure of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element modification needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
HierarchyElementComponentDeleteDirect(DimName, HierName, ConsolidatedElName, ElName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the consolidated element from which you want to delete a child. |
| HierName | The name of the hierarchy within the dimension. |
| ConsolidatedElName | The consolidated element from which you want to delete a child. |
| ElName | The name of the child element you want to delete. |

#### Example

```
HierarchyElementComponentDeleteDirect('Region', 'Western', 'Benelux', 'Belgium');
```

This example deletes the Belgium child from the Benelux consolidation in the Western hierarchy of the Region dimension.

## HierarchyElementDelete

HierarchyElementDelete deletes an element from a hierarchy.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyElementDelete(DimName, HierName, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension that contains the element you want to delete. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | The element you want to delete from the hierarchy. |

#### Example

```
HierarchyElementDelete('Region', 'Western', 'Belgium');
```

This example deletes the element Belgium from the Western hierarchy in the Region dimension.

## HierarchyElementDeleteDirect

HierarchyElementDeleteDirect deletes an element from a dimension by directly editing the dimension.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** Deleting an element deletes all cube data identified by that element.

The default means of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like DimensionElementDelete) are used in the Metadata procedure of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element modification needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
HierarchyElementDeleteDirect(DimName, HierName, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension that contains the element you want to delete. |
| HierName | The name of the hierarchy within the dimension. |

| Argument | Description |
|---|---|
| ElName | The element you want to delete. |

**Example**

```
HierarchyElementDeleteDirect('Region', 'Western', 'Belgium');
```

This example deletes the element Belgium from the Western hierarchy in the Region dimension.

## HierarchyElementExists

HierarchyElementExists determines whether a specific elements exists in a hierarchy on the server from which a TurboIntegrator process is executed. The function returns 1 if the elements exists in the hierarchy on the server, otherwise it returns 0.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyElementExists(DimName, HierName, ElemName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension that contains the element whose existence you want to confirm. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | The element you want to find in the hierarchy. |

**Example**

```
HierarchyElementExists('Region', 'Western', 'Belgium');
```

This example determines whether element Belgium from the Western hierarchy in the Region dimension exists on the server.

## HierarchyElementInsert

HierarchyElementInsert adds an element to a dimension. You can use this function to add numeric, string, or consolidated elements. You can't use this function in the Data or Epilog procedures of a TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyElementInsert(DimName, HierName, InsertionPoint, ElName, ElType);
```

| Argument | Description |
|---|---|
| DimName | The dimension to which you want to add a new element. |
| HierName | The name of the hierarchy within the dimension. |

| Argument | Description |
|---|---|
| InsertionPoint | An existing dimension element. The element being added to the dimension will be inserted immediately before this existing element. If this parameter is empty, the new element is added to the end of the dimension. |
| ElName | The name you want to assign to the new element. |
| ElType | The element type. There are three possible ElType values:<br>N - Signifies a numeric element.<br>S - Signifies a string element.<br>C - Signifies a consolidated element. |

**Example**

```
HierarchyElementInsert('Region', 'Western', 'Belgium', 'Netherlands','N');
```

This example adds the numeric element Netherlands to the Western hierarchy in the Region dimension. Netherland displays immediately before Belgium in the dimension definition.

## HierarchyElementInsertDirect

HierarchyElementInsertDirect adds an element to a dimension by directly editing the dimension. You can use this function to add numeric, string, or consolidated elements.

This function is valid in TM1 TurboIntegrator processes only.

The default means of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like HierarchyElementInsert) are used in the metadata tab of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element insertion needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
HierarchyElementInsertDirect(DimName, HierName, InsertionPoint, ElName, ElType);
```

| Argument | Description |
|---|---|
| DimName | The dimension to which you want to add a new element. |
| HierName | The name of the hierarchy within the dimension. |
| InsertionPoint | An existing dimension element. The element being added to the dimension will be inserted immediately before this existing element. If this parameter is empty, the new element is added to the end of the dimension.<br>Note that this function is optimized for the case where the InsertionPoint is passed as an empty string. |
| ElName | The name you want to assign to the new element. |
| ElType | The element type. There are three possible ElType values:<br>N - Signifies a numeric element.<br>S - Signifies a string element.<br>C - Signifies a consolidated element. |

**Example**

```
HierarchyElementInsertDirect('Region', 'Western', 'Belgium', 'Netherlands','N');
```

This example adds the numeric element Netherlands to the Western hierarchy in the Region dimension. Netherlands displays immediately before Belgium in the dimension definition.

## HierarchyElementPrincipalName

HierarchyElementPrincipalName returns the principal name of an element or element alias.

TurboIntegrator must use principal element names when updating dimensions; element aliases cannot be used. This function is therefore useful for determining principal element names while attempting to update a dimension when only element aliases are available to the TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyElementPrincipalName( DimName, HierName, ElName )
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension from which you want to retrieve a principal element name. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | An element name. ElName can be either an element alias or a principal element name. |

**Example**

If ElName is not in the currently saved version of DimName, the function returns ElName.

If ElName is in DimName, whether as an element alias or a principal element name, it returns the principal name of the element.

## HierarchyExists

HierarchyExists determines whether a specific hierarchy exists on the server from which a TurboIntegrator process is executed. The function returns 1 if the hierarchy exists on the server, otherwise it returns 0.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyExists(DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension that contains the hierarchy whose existence you want to confirm. |
| HierName | The name of the hierarchy within the dimension. |

**Example**

```
HierarchyExists('Region', 'Europe');
```

This example determines if the Europe hierarchy, in the Region dimension, exists on the server.

## HierarchyHasOrphanedLeaves

HierarchyHasOrphanedLeaves returns an integer that represents the number of members in the specified hierarchy that are not components of a parent member in that hierarchy (that is, orphaned leaves). If there are no members, the function returns 0.

This function is valid in TurboIntegrator processes only.

### Syntax

```
HierarchyHasOrphanedLeaves(DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension that contains the hierarchy being reviewed. |
| HierName | The name of the hierarchy you are reviewing for orphaned leaf members. |

**Example**

```
HierarchyHasOrphanedLeaves('Region', 'Europe');
```

This example determines if the Europe hierarchy, in the Region dimension, contains any orphaned leaves.

## HierarchySortOrder

HierarchySortOrder sets a sort type and sense for dimension elements and for components of consolidated elements within a dimension. The sort order defined by DimensionSortOrder determines how the subset displays in the Subset Editor.

DimensionSortOrder sets properties for a dimension; the dimension is not actually sorted until it is saved on the server.

This function is valid in TM1 TurboIntegrator processes only.

## Syntax

```
HierarchySortOrder(DimName, HierName, CompSortType, CompSortSense,ElSortType , ElSortSense);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension for which you want to set a sort order. |
| HierName | The name of the hierarchy within the dimension. |
| CompSortType | Defines how components of consolidated elements appear in the dimension. There are two CompSortType values:<br>ByInput - Retains the order in which components were originally inserted into consolidations.<br>ByName - Sorts components of consolidations by name. |
| CompSortSense | Defines the sort sense for components of consolidations. This is a required argument, but it applies only when the CompSortType is ByName. There are two possible CompSortSense values:<br>Ascending - Sorts components of consolidations in ascending alphabetical order.<br>Descending - Sorts components of consolidations in descending alphabetical order. |
| ElSortType | Defines a sort order for dimension elements. There are four possible ElSortType values:<br>ByInput - Retains the order in which elements were originally inserted into the dimension.<br>ByName - Sorts dimension elements by name.<br>ByLevel - Sorts dimension elements by level.<br>ByHierarchy - Sorts dimension elements by hierarchy. |

| Argument | Description |
|---|---|
| ElSortSense | Defines the sort sense for dimension elements. This is a required argument, but it applies only when the ElSortType is ByName or ByLevel. There are two possible ElSortSense values:<br>Ascending - Sorts dimension elements in ascending order, either alphabetically or by level.<br>Descending - Sorts dimension elements in descending order, either alphabetically or by level. |

**Example**

```
HierarchySortOrder ('Region', 'Europe', 'ByName', 'Descending','ByLevel', 'Ascending');
```

This example sets a sort order for the Europe hierarchy in the Region dimension. All dimension elements are sorted by level in ascending order, and any components of consolidations are sorted in descending alphabetical order.

## HierarchyTimeLastUpdated

HierarchyTimeLastUpdated indicates when a specified dimension hierarchy was last updated. The function returns a real number that represents the current day (including the hour, minute, second, and millisecond) since the beginning of the year 1900.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyTimeLastUpdated(dimension, hierarchy);
```

| Argument | Description |
|---|---|
| dimension | The name of the dimension. |
| hierarchy | The name of the hierarchy. |

**Example**

```
HierarchyTimeLastUpdated('Region', 'Europe');
```

This example returns information on when the Europe hierarchy of the Region dimension was last updated. If a value of 42548.<hours>.<minutes>.<milliseconds> is returned, you can divide 42548 by 365 to obtain (approximately) 116. When added to the started of 1900, the result is a current year of 2016.

## HierarchyTopElementInsert

HierarchyTopElementInsert creates a root element in a dimension. If the dimension already has a single root, then this element will not be created.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyTopElementInsert(DimName, HierName, InsertionPoint, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension for which you want to create a root element. |
| HierName | The name of the hierarchy within the dimension. |
| InsertionPoint | An existing dimension element. The root element being added to the dimension will be inserted immediately before this existing element. |
| ElName | The name you want to assign to the new root element. |

#### Example

```
HierarchyTopElementInsert('Region', 'Western', 'Netherlands', 'World');
```

This example adds the root element World to the Western hierarchy of the Region dimension. World is inserted displays immediately before Netherlands in the dimension definition.

## HierarchyTopElementInsertDirect

HierarchyTopElementInsertDirect creates a root element in a dimension by directly editing the dimension. If the dimension already has a single root, then this element will not be created.

This function is valid in TM1 TurboIntegrator processes only.

The default means of editing a dimension in TM1 is to use a whole-copy editing pattern. In that pattern, an editing copy of the dimension is created, edits are applied to the editing copy, then finally the actual dimension is rewritten using the editing copy as a template. TurboIntegrator supports whole-copy editing automatically whenever dimension editing TurboIntegrator functions (like HierarchyTopElementInsert) are used in the Metadata procedure of the process. TurboIntegrator automatically creates the editing copy and applies editing operations to it, then rewrites the actual dimension at the end of the Metadata procedure.

Direct edits are different in that no editing copy is involved. Instead, the operations are performed directly on the actual dimension. There are two different, specialized use cases for which this type of direct editing is intended:

- When the purpose of the TurboIntegrator process is to make a small change to a large dimension. In this case, direct editing will be more efficient because it avoids copying and completely rewriting the large dimension.
- When the purpose of the TurboIntegrator process is to load large volumes of data into a cube. In this case the process' Metadata procedure is deliberately kept empty, and any element modification needed to support data loading is performed using direct calls in the Data procedure. When the Metadata procedure is empty, the process skips an entire iteration over the external datasource, which can result in faster data loads.

### Syntax

```
HierarchyTopElementInsertDirect(DimName, HierName, InsertionPoint, ElName);
```

| Argument | Description |
|---|---|
| DimName | The dimension for which you want to create a root element. |
| HierName | The name of the hierarchy within the dimension. |
| InsertionPoint | An existing dimension element. The root element being added to the dimension will be inserted immediately before this existing element. |
| ElName | The name you want to assign to the new root element. |

**Example**

```
HierarchyTopElementInsertDirect('Region', 'Western', 'Netherlands', 'World');
```

This example adds the root element World to the Western hierarchy of the Region dimension. World is inserted displays immediately before Netherlands in the dimension definition.

### HierarchyUpdateDirect

HierarchyUpdateDirect performs a full rewrite of a hierarchy that has been subject to direct editing in a TurboIntegrator process, essentially compacting the memory footprint of the hierarchy.

A dimension that undergoes a series of direct-only edits (element deletions, in particular) will eventually use more memory than its fully-rewritten counterpart would. This function can optionally be used after directly editing a dimension with HierarchyElementInsertDirect, HierarchyElementDeleteDirect, HierarchyElementComponentAddDirect, HierarchyElementComponentDeleteDirect, and/or HierarchyTopElementInsertDirect. Calling HierarchyUpdateDirect incurs an initial full-copy memory cost, however it can be used to guarantee that the dimension is at its smallest possible memory footprint after processing is complete.

This function is valid in TM1 TurboIntegrator processes only.

#### Syntax

```
HierarchyUpdateDirect(DimName, HierName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension you want to rewrite. |
| HierName | The name of the hierarchy within the dimension. |

**Example**

```
HierarchyUpdateDirect('Region', 'Western');
```

This example rewrites the Western hierarchy of the Region dimension.

## ODBC TurboIntegrator Functions

These functions facilitate ODBC manipulation.

## ODBCClose

ODBCClose closes a connection to an ODBC data source.

This function is valid in TurboIntegrator processes only.

### Syntax

```
ODBCClose(Source);
```

| Argument | Description |
|---|---|
| Source | The name of an open ODBC data source. |

#### Example

```
ODBCClose('Accounting');
```

This example closes the connection to the Accounting ODBC source.

## ODBCOpen

ODBCOpen opens an ODBC data source for output.

This function is valid in TurboIntegrator processes only.

### Syntax

```
ODBCOpen(Source, ClientName, Password);
```

| Argument | Description |
|---|---|
| Source | An ODBC data source name. |
| ClientName | A valid client on the data source. |
| Password | A password for the ClientName. |

#### Example

```
ODBCOpen('Accounting', 'Jdoe', 'Bstone');
```

This example opens the Accounting ODBC data source for the Jdoe client using the password Bstone.

## ODBCOPENEx

ODBCOPENEx opens an ODBC data source for output specifying that the connection should be opened as a Unicode connection.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

Format is: ODBCOPENEx (dataset name, dataset client name, client password, (use-Unicode-interface flag) )

```
ODBCOpenEx(Source, ClientName, Password, UseUnicodeODBC);
```

| Argument | Description |
|---|---|
| Source | An ODBC data source name. |
| ClientName | A valid client on the data source. |
| Password | A password for the ClientName. |
| UseUnicodeODBC | Defines the type of Unicode connection to use. |

**Example**

```
ODBCOpenEx( TestTable, sa, , 1 );
```

```
chinese= ;
chinese = CHARW( 37123 );
fieldval = chinese | SomeNewText;
sql= Update TestTable set ForeName = N | fieldval |  WHERE CustomerId= 1
ODBCOUTPUT( Unicode, sql );
```

The result SQL statement looks like:

```
Update TestTable set ForeName = N?SomeNewText WHERE
CustomerId = 1
```

## ODBCOutput

ODBCOutput executes an SQL update query against an open ODBC data source. You should use the ODBCOpen function to open the data source before calling ODBCOutput, and use ODBCClose to close the data source before exiting the process.

This function is valid in TurboIntegrator processes only.

### Syntax

```
ODBCOutput(Source, SQLQuery, [SQLQuery2, SQLQuery3, ...]);
```

| Argument | Description |
|---|---|
| Source | The ODBC data source against which you want to run a query. |
| SQLQuery | An SQL query statement.<br>Though ODBCOutput was developed to update tables, it can be used to execute any SQL query on the data source.<br>In circumstances where the SQL query statement exceeds 255 characters, you should split the query into multiple SQLQuery arguments (SQLQuery2, SQLQuery3, etc.). This lets you create query statements that exceed the 255 character limit for TurboIntegrator arguments. When the ODBCOutput function is executed, all SQLQuery arguments are concatenated and the query is successfully executed. |

**Example**

```
ODBCOutput('Accounting', 'INSERT [CategoryID], [CategoryName]FROM Categories;');
```

This example executes the specified query against the Accounting data source.

## SetODBCUnicodeInterface

SetODBCUnicodeInterface sets whether the ODBC interface should use the Unicode wide functions or the regular single-byte character functions. Setting this function to 1 uses the wide character ODBC interface.

Some ODBC driver support either the older single-byte interface as well as a Unicode style 'wide-character' interface, where characters are passed and retrieved as 16-bit quantities. If the driver chosen does not support one or the other style, a flag is provided to force TurboIntegrator to use a particular style of interface.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
SetODBCUnicodeInterface=1
```

| Argument | Description |
|---|---|
| 1 | Use the wide character ODBC interface. |
| 0 | Use the single-byte interface. |

# Process Control TurboIntegrator Functions

These functions pertain to process control.

## ExecuteCommand

ExecuteCommand executes a command line during a process. You can use ExecuteCommand to run a desktop application, but not a service.

If you use ExecuteCommand to run an executable, the following conditions apply:

- If the CommandLine argument specifies only the name of a file to be executed, a Windows server looks for the file in both the server database directory and in the directory where Tm1s.exe resides. A UNIX server looks for the file only in the server database directory.
- If the CommandLine argument uses a relative path prefix, both the Windows and UNIX server attempt to locate the file in the server database directory only.
- On either the Microsoft Windows or UNIX server, you can pass an absolute path to the CommandLine argument to execute a file in any location..

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
ExecuteCommand(CommandLine, Wait);
```

| Argument | Description |
|---|---|
| CommandLine | The command line you want to execute. |
| Wait | Indicates if the process should wait for the command to complete execution before continuing to the next process statement. An argument value of 0 causes the process to proceed to the next statement without waiting for the command line to execute. An argument value of 1 causes the process to wait for the command line to successfully execute before proceeding to the next statement. |

## ExecuteProcess

ExecuteProcess lets you execute a TurboIntegrator process from within another process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ExecuteProcess(ProcessName, [ParamName1, ParamValue1,ParamName2, ParamValue2]);
```

| Argument | Description |
|---|---|
| ProcessName | The name of the process to be executed. This process must reside on the same server as the process from which ExecuteProcess is called.<br>If the process named by this argument cannot be found at runtime, the calling process is immediately terminated. (TurboIntegrator does not check for a valid ProcessName at compilation.) |
| ParamName | The name of an existing parameter of the process to be executed. This argument is required only if the process to be executed uses parameters. |
| ParamValue | A valid value for the ParamName parameter. If you specify a ParamName argument, you must specify a corresponding ParamValue.<br>The ParamName and ParamValue arguments must occur in ordered pairs, with the name of the parameter followed by the value. You must specify a ParamName and corresponding ParamValue for each parameter of the process to be executed. |

The parameter names passed in the ExecuteProcess function are matched at runtime against the parameter names specified in the process to be executed. If the passed names cannot be found in the parameter list of the process to be executed, a serious error results, causing the immediate termination of the process from which ExecuteProcess is called.

### Return Values

ExecuteProcess returns a real value that can be tested against one of the following return value functions:

| Function | Description |
|---|---|
| ProcessExitByChoreQuit() | indicates that the process exited due to execution of the ChoreQuit function |
| ProcessExitNormal() | indicates that the process executed normally |
| ProcessExitMinorError() | Indicates that the process executed successfully but encountered minor errors. |
| ProcessExitByQuit() | Indicates that the process exited because of an explicit "quit" command. |
| ProcessExitWithMessage() | Indicates that the process exited normally, with a message written to `tm1server.log`. |
| ProcessExitSeriousError() | Indicates that the process exited because of a serious error.<br>When a process exits because of a serious error, the process is immediately terminated and no further processing occurs. For example, if a serious error occurs in the Data procedure, the process immediately exits and the Epilog procedure is not executed. |
| ProcessExitOnInit() | Indicates that the process aborted during initialization. |
| ProcessExitByBreak() | Indicates that the process exited because it encountered a ProcessBreak function. |

**Example**

To record when a process called by ExecuteProcess fails because of a serious error, use code similar to the following:

```
return_value = ExecuteProcess('create_sales_cube');
ASCIIOutput('C:\temp\process_return_value.txt', 'Process exited
with serious errors at', TIME, 'on', TODAY);if(return_value = ProcessExitSeriousError() )
endif;
```

## GetProcessErrorFileDirectory

GetProcessErrorFileDirectory returns the full pathname, with trailing slash, of the directory where TurboIntegrator process error files are written. By default, all process error log files are written to the data directory of the server on which the process resides.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
GetProcessErrorFileDirectory;
```

### Arguments

None.

## GetProcessErrorFilename

GetProcessErrorFilename returns the name of the TurboIntegrator process error log file associated with a process. If the process has not yet generated an error log file, the function returns an empty (null) string.

**Important:** A process error log file is not generated until all statements in a given process tab (Prolog, Metadata, Data, or Epilog) have executed. Accordingly, you can use GetProcessErrorFilename to check if any previous tabs have generated an error log file, but you cannot use the function to determine if the current process tab causes errors to be written to a log file.

For example, by determining that GetProcessErrorFilename returns a non-null string in the Epilog tab, you can tell that errors were generated in the Prolog, Metadata, or Data tabs. However, you cannot use GetProcessErrorFilename in the Data tab to determine if the Data tab generates errors.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
GetProcessErrorFilename;
```

### Arguments

None.

## GetProcessName

GetProcessName returns as a string the name of the current process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
GetProcessName()
```

### Arguments

None.

```
Name = GetProcessName();
```

## If

The If statement allows a process to execute a statement or series of statements when a given expression is true. You can use arithmetic operators, logical operators, and comparison operators to construct an expression.

The TurboIntegrator If statement differs from the Rules IF function in that the TurboIntegrator statement can accept multiple ElseIf or Else statements to evaluate multiple expressions, while the Rules IF function can evaluate only one expression.

You can nest up to 20 If/ElseIf/Else statements in a TurboIntegrator process. If you exceed 20 nested If/ElseIf/Else statements, you will receive an error when attempting to save the process.

This function is valid in processes only.

### Syntax

```
If(expression);
statement1;
ElseIf(expression);
statement2;
ElseIf(expression);
```

```
statement3;
Else;
statement4;
EndIf;
```

## Arguments

None.

### Examples

```
If (x=5);
    ASCIIOutput('c:\temp\if.txt','x equals five');
ElseIf (x=1);
    ASCIIOutput ('c:\temp\if.txt', 'x equals one');
ElseIf (x=2);
    ASCIIOutput ('c:\temp\if.txt', 'x equals two');
ElseIf (x=3);
    ASCIIOutput ('c:\temp\if.txt', 'x equals three');
ElseIf (x=4);
    ASCIIOutput ('c:\temp\if.txt', 'x equals four');
Else;
    ASCIIOutput ('c:\temp\if.txt', 'x falls outside expected range');
EndIf;
```

This example evaluates the value of X. If X=5, the ASCIIOutput function is executed to write the string `x equals five` to c:\temp\if.txt. If X does not equal 5, the first ElseIf statement is evaluated. If X=1, the ASCIIOutput function is executed to write the string `x equals one` to `c:\temp\if.txt`. This processing continues until the EndIf is executed.

Simple If statements can also be constructed without the use of ElseIf, as in this example:

```
IF(expression);
    statement1;
ELSE;
    statement2;
ENDIF;
```

# ItemReject

ItemReject rejects a source record and places it in the error log, along with a specified error message.

This function is valid in TM1 TurboIntegrator processes only.

When ItemReject is executed in the Prolog of a process, any code following the ItemReject function in the Prolog is skipped and the process proceeds directly to the next procedure in the TurboIntgerator process.

## Syntax

```
ItemReject(ErrorString);
```

*Table 18. ItemReject arguments*

| Argument | Description |
|---|---|
| ErrorString | The error message you want written to the error log when a record is rejected. |

### Example

```
ItemReject(' Value outside of acceptable range.');
```

This example places a source record in the error log, along with the error message `Value outside of acceptable range.` when the source record contains a value that is beyond a defined range.

## ItemSkip

ItemSkip forces a process to skip the current data source item.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ItemSkip;
```

### Arguments

None.

## ProcessBreak

ProcessBreak stops processing source data and proceeds to the Epilog portion of a process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessBreak;
```

### Arguments

None.

## ProcessError

ProcessError causes an immediate termination of a process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessError;
```

### Arguments

None.

## ProcessExists

ProcessExists determines whether a specific TurboIntegrator process exists.

The ProcessExists function returns one of three possible values:

- If a TurboIntegrator process with the specified name does not exist, the function returns 0.
- If a process with the specified name does exist and is valid, the function returns 1.
- If a process with the specified name does exist, but has compilation errors, the function returns -1.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessExists(ProcessName);
```

| Argument | Description |
|---|---|
| ProcessName | The name of the process for which you are trying to determine status. |

## ProcessExitByChoreRollback

ProcessExitByChoreRollback initiates a chore rollback and exits with an error code. Similar to ChoreRollback, when used inside a TurboIntegrator process, this function throws out all pending edits and cancels further processing. An error message appears in the tm1server.log and tm1processorerrorXXX.log files.

When used in a single-commit mode chore, ProcessExitByChoreRollback throws out all pending edits from all previous processes and exits.

When used in a multi-commit mode chore, ProcessExitByChoreRollback throws out all pending edits from the current processes and then exits. Changes that have already been committed cannot be rolled back.

ProcessExitByChoreRollback returns the error code number.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessExitByChoreRollback;
```

### Arguments

None.

## ProcessExitByProcessRollback

ProcessExitByProcessRollback initiates a process rollback and exits with an error code. Similar to ProcessRollback, when used inside a TurboIntegrator process, this function throws out all pending edits and cancels further processing. An error message appears in the tm1server.log and tm1processorerrorXXX.log files.

When used in a single-commit mode chore, ProcessExitByProcessRollback throws out all pending edits from all previous processes and exits.

When used in a multi-commit mode chore, ProcessExitByProcessRollback throws out all pending edits from the current process and then exits. Changes that have already been committed cannot be rolled back.

ProcessExitByProcessRollback returns the error code number.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessExitByProcessRollback;
```

### Arguments

None.

## ProcessQuit

ProcessQuit terminates a TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessQuit;
```

### Arguments

None.

## ProcessRollback

ProcessRollback initiates a process rollback. When used inside a TurboIntegrator process, this function throws out all pending edits and cancels further processing. An error message appears in the tm1server.log and tm1processorerrorXXX.log files.

**Note:** In IBM Planning Analytics version 2.0.8 or later, when a TurboIntegrator process rolls back and restarts, the process is now represented in the `tm1server.log` file as three steps: starting, restarting because of lock contention or rollback, and then finishing.

An entry is added to the `tm1server.log` file that shows the TurboIntegrator process as restarting due to lock contention or rollback instead of just starting. This logging is enabled by default without setting any specific debug options.

When used in a single-commit mode chore, ProcessRollback throws out all pending edits from all previous processes and continues execution at the next process in the chore. If lock contention is encountered after the call to ProcessRollback, the entire chore is restarted.

When used in a multi-commit mode chore, ProcessRollback throws out all pending edits from the current process and then continues execution at the next process in the chore. Changes that have already been committed cannot be rolled back. If lock contention is encountered after the call to ProcessRollback, only the current process is restarted.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ProcessRollback;
```

### Arguments

None.

## RunProcess

RunProcess lets you run TurboIntegrator processes in parallel, each on its own thread that is managed by TM1 Server. This approach speeds up data load and other operations where TurboIntegrator processes are used to divide the work.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
RunProcess(ProcessName, [ParamName1, ParamValue1,ParamName2, ParamValue2]);
```

| Argument | Description |
|---|---|
| ProcessName | The name of the process to be run. This process must reside on the same server as the process from which RunProcess is called.<br>If the process named by this argument cannot be found at runtime, the calling process is immediately terminated. (TurboIntegrator does not check for a valid ProcessName at compilation.) |
| ParamName | The name of an existing parameter of the process to be run. This argument is required only if the process to be run uses parameters. |
| ParamValue | A valid value for the ParamName parameter. If you specify a ParamName argument, you must specify a corresponding ParamValue.<br>The ParamName and ParamValue arguments must occur in ordered pairs, with the name of the parameter followed by the value. You must specify a ParamName and corresponding ParamValue for each parameter of the process to be run. |

The parameter names passed in the RunProcess function are matched at runtime against the parameter names specified in the process to be run. If the passed names cannot be found in the parameter list of the process to be run, a serious error results, causing the immediate termination of the process from which RunProcess is called.

### Return values

RunProcess returns a string. The string is the JobID, or an empty string if an error occurs.

## Sleep

Use this function to pause, or 'sleep' a process for a specified interval, expressed in milliseconds.

### Syntax

```
Sleep(ms);
```

| Argument | Description |
|---|---|
| ms | The number of milliseconds that you want the process to pause. |

#### Example

```
# Pause the process for 3 seconds
sleep ( 3000 );
```

## Synchronized

Synchronized is used in a TurboIntegrator script to force serial execution of a designated set of TurboIntegrator processes.

This function is valid in TurboIntegrator processes only.

## Syntax

The Synchronized function uses the following syntax.

```
Synchronized (lockName, nonBlocking);
```

Synchronized takes a single required parameter that is a user-defined name for a lock object. This lock object name can be used in multiple TurboIntegrator processes in order to serialize their execution as a group.

*Table 19. Synchronized arguments*

| Argument | Description |
|---|---|
| `lockName` | The user-defined name of a lock object on which to synchronize. Names are case-insensitive and embedded spaces are ignored. Names may not exceed 1023 characters in length.<br>String/Yes/None |
| `nonBlocking` | Optional. If set to 1, this function does not block if the lock object is already in use. Instead, the function returns a value of 1. This allows the calling process to take alternative action, such as exiting early by using the **`ProcessQuit`** function.<br>If set to 0 or not defined, the function blocks normally if the lock object is already in use. |

## Semantics

A TurboIntegrator process may make any number of calls to **Synchronized**, with any number of lock objects. Serializing is effective from the time synchronized is called, until the containing transaction completes.

For example, if **Synchronized** is called from a subprocess (Ps) of primary process (Pp) or primary chore (Cp), the Lock Object is released when Pp or Cp completes. The exception is that a SaveDataAll (SDA) prematurely ends a transaction mid-process execution; this applies to Lock Objects as well.

The **Synchronized** call can be placed anywhere within a TurboIntegrator script, but serialization applies to the entire TurboIntegrator process when it is encountered.

Consider a TurboIntegrator process with a **Synchronized** call somewhere in the middle of its script, and an operation O1 preceding that call. Two instances of this TurboIntegrator process may start at the same time. It is possible for one instance to run to completion, including its call to **Synchronized**, before the second instance reaches its **Synchronized** call. In this case, the two processes appear to the user to have run concurrently. If, instead, the second process does reach its synchronized() call before the first completes, it will undo any work it had done (O1) and wait for the first to complete. In this case, the two processes appear to the user to have serialized.

To avoid such confusion, and to optimize the use of **Synchronized**, it is recommended (but not enforced) that **Synchronized** calls be the first statements of a TurboIntegrator process.

### Example

Consider that TurboIntegrator process P needs to update two cubes, Cube_1 and Cube_2.

Other TurboIntegrator processes may also need to update Cube_1 or Cube_2.

To cause all TurboIntegrator processes that will update Cube_1 or Cube_2, to run one at a time, P could call **Synchronized** in this manner:

```
sCube_1='Cube_1';
sCube_2='Cube_2';
sE1='Elm1';
```

```
sE2='Elm2';
sE4='Units';
sE5='Price';

Synchronized( sCube_1 );
Synchronized( sCube_2 );

CellPutn( 111, sCube_1, sE1, sE2 );
CellPutn( 9.99, sCube_2, sE4, sE5 );

# ...
```

Other TurboIntegrator processes that will update Cube_1 or Cube_2 must also call `Synchronized( sCube_1 )` and/or `Synchronized( sCube_2 )` in a similar way.

In this example, the two lock objects' names were chosen to be the same as the cubes' names. But a lock object's name does not have to be the same as other objects (cubes, dimensions, subsets).

### While

The While statement allows a process to repeat a series of statements while a given condition is true. While statements can be nested.

This function is valid in TM1 TurboIntegrator processes only.

#### Syntax

```
WHILE(logical expression);

statement1;

statement2;

...

statement n;

END;
```

**Note:** All WHILE statements must conclude with an END statement.

#### Arguments

None.

## Rules Management TurboIntegrator Functions

These functions facilitate rules management.

### CubeProcessFeeders

CubeProcessFeeders reprocesses all feeders in the rules for a specified cube.

This function reprocesses all feeders in the rules for a specified cube. The feeders are normally reprocess automatically when a rule file edit is saved, however, if the data changes, and those data changes will change some conditional feeders, this function will need to be called to get those conditional feeders re-evaluated.

This function is valid in TM1 TurboIntegrator processes only.

#### Syntax

```
CubeProcessFeeders(CubeName);
```

| Argument | Description |
|---|---|
| CubeName | The cube for which you want to reprocess feeders. |

**Example**

```
CubeProcessFeeders('2003sales');
```

This example reprocesses all feeders in the rules for the 2003sales cube.

## CubeRuleAppend

CubeRuleAppend appends a single line of rule text to a Planning Analytics cube rule.

Essentially, this function adds a single line of text to a rule (.rux) file. The line of text is typically a rule statement, but can also be a comment. If there is no rule associated with the cube at the time this function is executed, a new rule is created, containing only the passed line.

This function is valid only in Planning Analytics processes.

### Syntax

```
CubeRuleAppend(CubeName, RuleText, IsCalculationRule);
```

| Argument | Description |
|---|---|
| CubeName | The name of the cube associated with the rule to which you want to append a line of text. |
| RuleText | The single line of text you want to append to the rule.<br>The entire line of text you add must be enclosed in single quotes and must adhere to rules syntax conventions.<br>If the line of text includes any element references, the element names must be enclosed in double single quotes to escape the single quotes that normally enclose element names. For example, a reference to an element named CL3 must be specified as `[''CL3'']`.<br>The following are examples of valid lines of text you might append to a rule:<br>`'[''CL3''] = [''CL4''] + [''Trial''];'`<br>`'skipcheck;'`<br>`'[''Trial''] => [''CL3''];'` |

| Argument | Description |
|---|---|
| IsCalculationRule | The IsCalculationRule parameter declares whether the line should be inserted just before any feeder section that might exist in the cube rule. If the IsCalculationRule parameter is omitted, or passed as 0.0, then the new line will simply be appended to the end of the rule.<br>Because rule (.rux) files consist of a calculation section followed by an optional feeder section, any appended lines that are calculation rule statements (or corresponding comments) should use a 1.0 for this argument to ensure that the new line is inserted in at the appropriate location in the rule file. |

**Examples**

```
CubeRuleAppend( 'MyCube', ''['']CL3'''] = ''['']CL4'''] + ''['']Trial''''];', 1.0 );
```

This example inserts the calculation statement `['CL3'] = ['CL4'] + ['Trial'];` at the end of the calculation section of the rule for the MyCube cube.

```
CubeRuleAppend( 'MyCube', ''['']Trial'''] => ''['']CL3''''];', 0.0 );
```

This example inserts the feeder statement `['Trial'] => ['CL3'];` at the end of the rule for the MyCube cube.

## CubeRuleDestroy

CubeRuleDestroy deletes any rule that exists for a specified cube.

This function is valid in TM1 TurboIntegrator processes only.

## Syntax

```
CubeRuleDestroy(CubeName);
```

| Argument | Description |
|---|---|
| CubeName | The name of the cube associated with the rule that you want to delete |

**Example**

```
CubeRuleDestroy('SalesProjections');
```

This example deletes the rule for the SalesProjectionscube.

## CubeRuleGet

CubeRuleGet retrieves a specified cube rule as a single string. This function is valid only in Planning Analytics processes.

### Syntax

```
CubeRuleGet(RuleName);
```

| Argument | Description |
|---|---|
| RuleName | The name of the rule that you want to retrieve. You do not need to specify the .rux file extension.<br>The rule must exist on the database where the process is executed. |

**Example**

`CubeRuleGet('RevenueRule');`

The `RevenueRule` contains these three lines:

```
['Gross Margin %']=c: (['Gross Margin']\['Gross Revenue'])*100;
['Unit Price'] = c: ['Gross Revenue']\['Units Sold'];
['Unit Cost'] = c: ['Cost of Sales']\['Units Sold'];
```

The example returns this single string:

```
'['Gross Margin %']=c: (['Gross Margin']\['Gross Revenue'])*100; ['Unit Price']
= c: ['Gross Revenue']\['Units Sold']; ['Unit Cost'] = c: ['Cost of Sales']\
['Units Sold'];'
```

## CubeRuleSet

CubeRuleSet replaces the content of a cube rule with a specified string. This function is valid only in Planning Analytics processes.

### Syntax

```
CubeRuleSet(RuleName, RuleString);
```

| Argument | Description |
|---|---|
| RuleName | The name of the cube rule for which you want to replace content. You do not need to specify the .rux file extension.<br>If the rule does not exist on the database where the process is executed, a new rule is created upon execution of this function, containing the rule statements included in the RuleString. |

| Argument | Description |
|---|---|
| RuleString | A single text string containing the rules statements that you want to write to the cube rule.<br>The entire string must be enclosed in single quotes and must adhere to rules syntax conventions.<br>If the string includes any member references, the member names must be enclosed in double single quotes to escape the single quotes that normally enclose member names in a rule. For example, a reference to a member named CL3 must be specified as `[''CL3'']` |

**Example**

```
CubeRuleSet('RevenueRule', '[''Gross Margin %'']=c: ([''Gross Margin'']\
[''Gross Revenue''])*100; [''Unit Price''] = c: [''Gross Revenue'']\[''Units
Sold'']; [''Unit Cost''] = c: [''Cost of Sales'']\[''Units Sold''];');
```

Note that all member references in the RuleString are enclosed in double single quotes to escape the single quotes that normally enclose member names.

This example replaces any existing content in `RevenueRule` with these three lines:

```
['Gross Margin %']=c: (['Gross Margin']\['Gross Revenue'])*100;
['Unit Price'] = c: ['Gross Revenue']\['Units Sold'];
['Unit Cost'] = c: ['Cost of Sales']\['Units Sold'];
```

If `RevenueRule` does not already exist, it is created on the database where the process is executed.

## DeleteAllPersistentFeeders

DeleteAllPersistentFeeders deletes any .feeder files that have persisted. When this function is used, all cubes are marked as "do not save feeders" so a subsequent SaveData will not persist feeders which means all feeders will be re-calculated on a server re-start.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DeleteAllPersistentFeeders;
```

### Arguments

None.

## ForceSkipCheck

ForceSkipCheck forces the query to perform as if the cube had a SKIPCHECK in the rules.

This function is valid in TM1 TurboIntegrator processes only.

This means that the query will process only values actually in the cube, as opposed to (the no SKIPCHECK case) where every possible cell would be enumerated looking for values. This function sets the state of the view query to select only values in the cube. The function must be added to the Prolog section of the TurboIntegrator process. By placing the ForceSkipCheck() in the Prolog it effects the entire view query of data elements to follow.

### Syntax

```
ForceSkipCheck()
```

### Arguments

None.

## RuleLoadFromFile

RuleLoadFromFile creates a TM1 rule for a specified cube from a text file. Each rule statement must end with a semi-colon (;) and comments must be prefixed with the # character. If a rule already exists for the specified cube, the rule is overwritten by the rule created by RuleLoadFromFile.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
RuleLoadFromFile(Cube, TextFile);
```

| Argument | Description |
|---|---|
| **Cube** | The name of the cube for which you want to create a rule. |
| **TextFile** | The name of the text file from which you want to create a rule.<br>You can specify the full path to this file, including file name and extension. (Refer to Example 1.)<br>If you specify only the file name and extension, TurboIntegrator looks for the file in the server's data directory.<br>If you do not specify a file extension, TurboIntegrator assumes the `.rux` extension by default. (Refer to Example 2.) |

If you leave the TextFile argument empty, TurboIntegrator looks for a source file with the same name as the cube (but with a .rux extension) in the server's data directory. (Refer to Example 3.)

#### Example 1:

The following example uses the contents of the cuberule.txt file in the C:\temp directory to create a rule for the Sales cube:

```
RuleLoadFromFile('Sales', 'C:\temp\cuberule.txt');
```

#### Example 2:

This example creates a rule for the Sales cube using the file named cuberule.rux in the server's data directory:

```
RuleLoadFromFile('Sales', 'cuberule');
```

**Example 3:**

This example creates a rule for the Sales cube using the file named Sales.rux in the server's data directory:

```
RuleLoadFromFile('Sales', ' ');
```

## RuleLoadFromFileEx

RuleLoadFromFileEx creates a Planning Analytics rule for a specified cube from a text file using a specified character set. Each rule statement in the text file must end with a semi-colon (;) and comments must be prefixed with the # character. If a rule already exists for the specified cube, the rule is overwritten by the rule created by RuleLoadFromFileEx.

This function is valid in TurboIntegrator processes only.

This function is similar to the “RuleLoadFromFile” on page 1164 function, but provides the ability to specify the character encoding used in the text file.

### Syntax

```
RuleLoadFromFileEx(Cube, TextFile, CharacterSet);
```

| Argument | Description |
|---|---|
| **Cube** | The name of the cube for which you want to create a rule. |
| **TextFile** | The name of the text file from which you want to create a rule.<br>You can specify the full path to this file, including file name and extension.<br>If you specify only the file name and extension, TurboIntegrator looks for the file in the server's data directory.<br>If you do not specify a file extension, TurboIntegrator assumes the .rux extension by default. |
| **CharacterSet** | The character encoding used in the **TextFile**. For a list of valid values, refer to this table. |

**Example**

This example uses the contents of the `cuberule.txt` file in the `C:\temp` directory to create a rule for the Sales cube, specifying that `cuberule.txt` uses TM1CS_EUC_CN (EUC Simplified Chinese) character encoding.

```
RuleLoadFromFileEx('Sales', 'C:\temp\cuberule.txt', 'TM1CS_EUC_CN');
```

# Sandbox Functions

These functions are used with sandboxes.

## GetUseActiveSandboxProperty

GetUseActiveSandboxProperty returns a Boolean value that indicates whether a process reads and writes data to the base data or to the user's active sandbox.

This function is valid in TM1 TurboIntegrator processes only.

The default is for processes to read and write to the base data.

- If the return is 0, the process is currently reading and writing to the base data.
- If the return is 1, the process is currently reading and writing to the active sandbox.

**Note:** This function returns the permanent value for this property as set in the Architect / Server Explorer user interface unless you used the SetUseActiveSandboxProperty function in the process. In that case, the value for this property is determined by the value that was last set with the SetUseActiveSandboxProperty function.

### Syntax

```
GetUseActiveSandboxProperty()
```

### Arguments

None.

#### Example

```
return_value = GetUseActiveSandboxProperty();
```

This example will return a Boolean value indicating whether the process is currently reading and writing cube data to the active sandbox or to the base data.

## ServerActiveSandboxGet

ServerActiveSandboxGet returns the name of the user's active sandbox. If the user has no active sandbox, an empty string is returned. Because chores run in the context of a special admin user, and can have no active sandbox, this function always returns an empty string when executed using a chore.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerActiveSandboxGet();
```

### Arguments

None.

#### Example

```
return_value = ServerActiveSandboxGet();
```

This example will return the active sandbox of the user executing the TI process in which the function call is made.

## ServerActiveSandboxSet

ServerActiveSandboxSet sets the active sandbox of the executing user. An empty string is used to clear the executing user's active sandbox. This function throws an error if the executing user does not own a sandbox with the passed name.

Because chores run in the context of a special admin user, and can have no active sandbox, this function always throws an error when executed using a chore.

**Note:** For a TurboIntegrator process to read and write values in the context of the executing user's active sandbox, the UseActiveSandbox property must be set. See “GetUseActiveSandboxProperty” on page 1166 and “SetUseActiveSandboxProperty” on page 1175.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerActiveSandboxSet(SandboxName)
```

| Argument | Description |
|---|---|
| SandboxName | A string value. The name of a sandbox owned by the executing user. |

#### Example: Set the executing user's active sandbox to "Best case"

```
ServerActiveSandboxSet('Best case');
```

#### Example: Clear the executing user's active sandbox and set context back to the base data

```
ServerActiveSandboxSet('');
```

## ServerSandboxClone

ServerSandboxClone clones an existing sandbox into a new sandbox.

Sandboxes are private workspaces in which a user can enter and store data values separate from TM1 base data. Sandboxes are stored on disk and, when in use, in memory.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerSandboxClone(sandboxName,newSandboxName );
```

| Argument | Description |
|---|---|
| sandboxName | A string value. The name of a sandbox owned by the executing user. |
| newSandboxName | A string value. The name of a sandbox to be created as a clone of *sandboxName*. |

#### Example

```
ServerSandboxClone( 'Best case', 'Second best case');
```

## ServerSandboxCreate

ServerSandboxCreate creates a new sandbox.

Sandboxes are private workspaces in which a user can enter and store data values separate from TM1 base data. Sandboxes are stored on disk and, when in use, in memory.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerSandboxCreate( sandboxName );
```

| Argument | Description |
|---|---|
| sandboxName | A string value. The name of a sandbox to be created. |

#### Example

```
ServerSandboxCreate( 'My sandbox' );
```

## ServerSandboxesDelete

ServerSandboxesDelete allows administrators to discard user sandboxes that match certain criteria.

Sandboxes are private workspaces in which a user can enter and store data values separate from TM1 base data. Sandboxes are stored on disk and, when in use, in memory.

This function operates server side and is available through TurboIntegrator and the API function ServerSandboxesDelete. Using this feature in a TurboIntegrator process, administrators can schedule maintenance using automated chores.

This function is valid in TM1 TurboIntegrator processes only.

### Description

This function uses a "predicate" to describe the sandbox being deleted. A predicate can be read as: "Delete sandboxes whose *attribute* is *condition value*."

For example: "Delete sandboxes whose size is greater than 10 MB." In this example, the attribute is the "size" of the sandbox, the condition is "greater than", and the value is "10 MB".

There are two optional delimiter character parameters to the TurboIntegrator function. Because a sandbox has no restrictions on which characters can be used in their name, administrators can supply their own "safe" delimiter when needed.

For example, `ServerSandboxesDelete( 'client:=:Admin, name:=:best case scenario' );"`

In the following example, the colon character is used in the sandbox name ("best::case::scenario") so another delimiter is needed:

```
ServerSandboxesDelete( 'client|=|Admin# name|=|best::case::scenario', '|', '#' );"
```

**Note:** The exact syntax of a predicate differs between the TurbIntegrator and API forms of this function.

### Syntax

```
ServerSandboxesDelete(string,string,string)
```

| Argument | Description |
|---|---|
| Predicates | The name of the process to be executed. This process must reside on the same server as the process from which RunProcess is called.<br>Required<br>String<br>No default<br>An arbitrary length list of predicates. Each predicate is a string containing three tokens. The first token indicates an attribute of a sandbox. The second indicates a condition, for example ">" or "=". The third token is a possible value of the attribute on which sandboxes should be conditionally filtered. The entire string may not exceed 10,000 characters in length. |
| PredicateDelimiter | Optional<br>String<br>default is : (colon)<br>Optional delimiter character.<br>The string may not exceed 1 character in length. |
| PredicateListDelimiter | Optional<br>String<br>default is , (comma)<br>Optional delimiter character.<br>The string may not exceed 1 character in length. |

## Filter Attributes

Filter attributes are properties of a sandbox on which it can be conditionally matched. Attribute names and their corresponding valid conditions are case insensitive and ignore embedded whitespace. For example, the following two calls are both valid:

```
ServerSandboxesDelete( 'client:=:Admin' );
ServerSandboxesDelete( 'C L I E N T : = :Admin' );
```

*Table 20. Filter Attributes*

| Attribute | Description | Valid Conditions | Value Type |
|---|---|---|---|
| UpdateDate | Timestamp of the last write action performed in the sandbox. | <, =, >. | Timestamp in international standard format, i.e. YYYY-MM-DD. Days are the most granular units. |
| AccessDate | Timestamp of the last unload of a sandbox. | <, =, >. | Timestamp in international standard format, i.e. YYYY-MM-DD. Days are the most granular units. |
| CreationDate | Timestamp of the creation of a sandbox. | <, =, >. | Timestamp in international standard format, i.e. YYYY-MM-DD. Days are the most granular units. |

| Table 20. Filter Attributes (continued) | | | |
|---|---|---|---|
| **Attribute** | **Description** | **Valid Conditions** | **Value Type** |
| Size | The in-memory size of a sandbox. | <, =, >. | Size following log4cxx's conversion rules (see configuration parameter AuditLogMaxTemp FileSize) For example, 10 MB. Kilobytes are the most granular units. |
| Name | The name of a sandbox. | =, containing. | String. |
| Client | The owning client of a sandbox. | =. | String. |
| Group | A group of which the owning client of a sandbox is a member. | =. | String. |

## Logging and Returns

Sandbox deletion is logged using the preexisting audit logging functionality. Additionally, a more detailed report of the effects of sandbox administration is included in the debug log (tm1server.log) at INFO level. This report will include the list of affected sandboxes, as well as some of their attributes, and any errors encountered.

ServerSandboxesDelete returns only a success or failure status.

## Semantics

### Predicate List

Multiple predicates passed in a single call to ServerSandboxesDelete are conjunctive. In other words, for a sandbox to match the passed criteria, all predicates must be true. Multiple calls to ServerSandboxesDelete can be used to achieve disjunctive behavior. Only one occurrence of each attribute is allowed per call to ServerSandboxesDelete. For example, passing client twice is invalid as a sandbox has only one owning client. When multiple occurrences of an attribute are detected, a warning displays in the detailed report, however, the operation will not abort in failure. In such a case, the predicates are tested as with any other query, but the results set is always empty.

### Locking

To avoid massive locking issues, ServerSandboxesDelete looks at the sandboxes of a client as a point-in-time snapshot and then, when possible, release any locks that would ensure a serializable transaction. Because of this behavior, once a client is "passed" in the iteration of all clients, a sandbox matching the filter criteria may be added to that client before the maintenance transaction completes. This behavior is similar to the behavior that occurs when a sandbox is added to the client immediately after the transaction completes.

### Scope

Members of the ADMIN (super-user) and the DataAdmin groups will have access to all sandboxes of all clients. They must explicitly specify the client attribute to limit the scope of their call to ServerSandboxesDelete to only their own sandboxes. All other users have access to only their own sandboxes; if they specify a different client, or a group to which they do not belong, the function will abort in failure and return a privilege error.

### In-Use Sandboxes

When a sandbox meets the criteria for deletion, but is currently in use, that sandbox will not be deleted. An entry will appear in the debug log info-level report indicating the occurrence.

### Access and Update Dates

Date attributes can be matches with, at most, day granularity. Because of this restriction, recording of these attributes is correspondingly granular. Last Update Date is not updated on individual cell writes. Instead, the system records the unload date of a sandbox that has had something written to it while it was loaded in memory. For such sandboxes, Last Access Date and Last Update Date will be the same. Only Last Access Date is updated on the unloading of a sandbox from memory. Also, because in-memory sandboxes are not subject to ServerSandboxesDelete, Last Access Date is not updated when a sandbox is loaded into memory.

For example, consider the follow usage scenario:

*Table 21. Last Access Day Example*

| Day | Time | Action |
|---|---|---|
| 1 | 1 | Load Sandbox S |
| 1 | 2 | Write 1 |
| 2 | 3 | Read 1 |
| 2 | 4 | Unload Sandbox |

A user is working with sandbox over the course of two days (perhaps for a much shorter period encompassing the day change.) At time 4, when the sandbox is unloaded, Last Update Date is set to 2, rather than 1 where the last update actually occurred. Last Access Date is also set to 2 at time 4 in this case. If Write1 were instead a read, only Last Access Date would be set to 2, while Last Update Date wouldn't be changed.

### Example

```
ServerSandboxesDelete( 'client:=:Admin, name:=:best case scenario' );
```

## ServerSandboxDiscardAllChanges

ServerSandboxDiscardAllChanges discards all changes in an existing sandbox.

This function is valid in TM1 TurboIntegrator processes only.

## Syntax

```
ServerSandboxDiscardAllChanges( sandboxName );
```

| Argument | Description |
|---|---|
| sandboxName | A string value. The name of a sandbox owned by the executing user. |

### Example

```
ServerSandboxDiscardAllChanges( 'MySandbox' );
```

## ServerSandboxMerge

ServerSandboxMerge merges a source sandbox into an existing target sandbox. If the target sandbox is not specified, the source sandbox is merged into base.

Sandboxes are private workspaces in which a user can enter and store data values separate from TM1 base data. Sandboxes are stored on disk and, when in use, in memory.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerSandboxMerge( src, tgt, conflictRes, waitForLocks);
```

| Argument | Description |
|---|---|
| src | The name of the source sandbox owned by the executing user to be merged with the *<tgt>* sandbox.<br>The *<src>* sandbox is not changed.<br>Required<br>String |
| tgt | The name of a sandbox owned by the executing user to be merged with the *<src>* sandbox.<br>The *<tgt>* sandbox is updated.<br>If *<tgt>* is blank, you are merging *<src>* with base data and updating base.<br>Required. To leave this parameter blank, use 2 concatenated single quotes: `''`.<br>String |
| conflictRes | The *<conflictRes>* parameter is ignored.<br>Optional<br>Numeric |
| waitForLocks | The *<waitForlocks>* parameter is an integer that indicates whether to wait for locks to guarantee serialization.<br>1 means wait for locks, catch any conflict exceptions, and retry instead of allowing the chore or process to roll back.<br>0 means do not wait for locks. Allow exceptions that cause rollback.<br>Optional<br>Numeric |

#### Example

Merge `mySandbox` to base.

```
ServerSandboxMerge(mySandbox, '');
```

## ServerSandboxExists

ServerSandboxExists tests for the existence of the passed sandbox. 1 is returned when the passed sandbox exists, 0 otherwise.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerSandboxExists( sandboxname )
```

or

```
ServerSandboxExists( sandboxname , username )
```

### Arguments

The name of the sandbox whose existence is being tested. ServerSandboxExists takes an optional string parameter, the owning client's name. The calling client can use the optional parameter to specify a client other than themselves if the calling client has the appropriate privileges. A privilege error will result if the specified client is not the executing client and the executing client is not a member of the DataAdmin or ADMIN groups. If the optional parameter is not used, the active client's sandboxes are the subject.

#### Example

The following snippet shows how the ServerSandboxExists, ServerSandboxGet, and ServerSandboxListCountGet functions can be used to iterate the sandboxes of user called User1 and output those sandboxes to a text file. The TurboIntegrator process would successfully execute for members of the Admin or Data Admin groups and for user called User1. The TurboIntegrator process would fail with a privilege error for any other users.

```
SandboxIndex = 1;
NumSandboxes = ServerSandboxListCountGet( 'User1' );

WHILE( SandboxIndex <= NumSandboxes );

   SandboxName = ServerSandboxGet( SandboxIndex, 'User1' );

   IF( ServerSandboxExists( SandboxName, 'User1' ) = 1 );

      ASCIIOUTPUT( 'C:\User1Sandboxes.txt', SandboxName );

   ENDIF;

   SandboxIndex = SandboxIndex + 1;

END;
```

## ServerSandboxGet

ServerSandboxGet returns the name of the sandbox identified by the number *N*, where *N* is the parameter entered.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerSandboxGet( index )
```

or

```
ServerSandboxGet( index, username)
```

### Arguments

The index of the requested sandbox in the user's sandbox collection. The index space will be contiguous, so the first occurrence of an empty string return can be used to break iteration. Also, deleting a sandbox will alter the indexes of any sandboxes that follow that sandbox in the list.

ServerSandboxGet takes an optional string parameter, the owning client's name. The calling client can use the optional parameter to specify a client other than themselves if the calling client has the appropriate privileges. A privilege error will result if the specified client is not the executing client and the executing client is not a member of the DataAdmin or ADMIN groups. If the optional parameter is not used, the active client's sandboxes are the subject.

#### Example

The following snippet shows how the ServerSandboxExists, ServerSandboxGet, and ServerSandboxListCountGet functions can be used to iterate the sandboxes of user called User1 and output those sandboxes to a text file. The TurboIntegrator process would successfully execute for members of the Admin or Data Admin groups and for user called User1. The TurboIntegrator process would fail with a privilege error for any other users.

```
SandboxIndex = 1;
NumSandboxes = ServerSandboxListCountGet( 'User1' );

WHILE( SandboxIndex <= NumSandboxes );

   SandboxName = ServerSandboxGet( SandboxIndex, 'User1' );

   IF( ServerSandboxExists( SandboxName, 'User1' ) = 1 );

      ASCIIOUTPUT( 'C:\User1Sandboxes.txt', SandboxName );

   ENDIF;

   SandboxIndex = SandboxIndex + 1;

END;
```

## ServerSandboxListCountGet

ServerSandboxListCountGet returns the count of sandboxes as a number.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerSandboxListCountGet()
```

or

```
ServerSandboxListCountGet( username )
```

### Arguments

ServerSandboxListCountGet takes an optional string parameter, the owning client's name. The calling client can use the optional parameter to specify a client other than themselves if the calling client has the appropriate privileges. A privilege error will result if the specified client is not the executing client and the executing client is not a member of the DataAdmin or ADMIN groups. If the optional parameter is not used, the active client's sandboxes are the subject.

#### Example

The following snippet shows how the ServerSandboxExists, ServerSandboxGet, and ServerSandboxListCountGet functions can be used to iterate the sandboxes of user called User1 and output those sandboxes to a text file. The TurboIntegrator process would successfully execute for members of the Admin or Data Admin groups and for user called User1. The TurboIntegrator process would fail with a privilege error for any other users.

```
SandboxIndex = 1;
NumSandboxes = ServerSandboxListCountGet( 'User1' );
```

```
WHILE( SandboxIndex <= NumSandboxes );
   SandboxName = ServerSandboxGet( SandboxIndex, 'User1' );
   IF( ServerSandboxExists( SandboxName, 'User1' ) = 1 );
      ASCIIOUTPUT( 'C:\User1Sandboxes.txt', SandboxName );
   ENDIF;
   SandboxIndex = SandboxIndex + 1;
END;
```

## SetUseActiveSandboxProperty

SetUseActiveSandboxProperty controls whether a process reads and writes cube data to the base data or to the user's active sandbox. The default is for processes to read and write to the base data.

The scope of this function applies only to the current running process and temporarily overrides the permanent value for this property that is set in the Architect / Server Explorer user interface.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SetUseActiveSandboxProperty(PropertyValue)
```

| Argument | Description |
|---|---|
| PropertyValue | A Boolean value that indicates whether the process should use the active sandbox context when reading and writing cube data.<br>If PropertyValue = 0, the process will disregard the active sandbox context and read/write to the base data.<br>If PropertyValue = 1, the process will read/write cube data to the active sandbox. |

#### Example

```
SetUseActiveSandboxProperty(1);
```

This example will cause the process to read/write cube data to the active sandbox for the rest of this execution.

# Security TurboIntegrator Functions

These functions pertain to security.

## AddClient

AddClient creates a new client on the server. Changes applied through the AddClient functions do not take effect until the Metadata procedure in a process is completed. This function, like all functions that update metadata, should not be used in the Data or Epilog tabs of a process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AddClient(ClientName);
```

| Argument | Description |
|---|---|
| ClientName | The name of the client you want to add to the server.<br>The client name is limited to 255 characters/bytes. |

#### Example

```
AddClient('Brian');
```

This example adds the client Brian to the server.

## AddGroup

AddGroup creates a new user group on the server. Changes applied through the AddGroup function do not take effect until the Metadata procedure in a process is completed. This function, like all functions that update metadata, should not be used in the Data or Epilog tabs of a process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AddGroup(GroupName);
```

| Argument | Description |
|---|---|
| GroupName | The name of the group you want to create. |

#### Example

```
AddGroup('Finance');
```

This function adds the Finance user group to the server.

## AssignClientToGroup

AssignClientToGroup assigns an existing client on a server to an existing user group. This function assigns an existing client on a server to an existing user group.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
AssignClientToGroup(ClientName, GroupName);
```

| Argument | Description |
|---|---|
| ClientName | The name of the client you want to assign to a group. |
| GroupName | The group to which you want to assign the client. |

**Example**

```
AssignClientToGroup('Brian', 'Finance');
```

This example assigns the existing client Brian to the existing user group Finance.

## AssignClientPassword

AssignClientPassword assigns a password to an existing client on a server. AssignClientPassword returns 1 if the password assignment is successful and returns 0 if the assignment fails.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
AssignClientPassword (ClientName, Password);
```

| Argument | Description |
|---|---|
| ClientName | The name of the client for which you want to assign a password. |
| Password | The password you want to assign to the client. When assigning a password, use plain text. TM1 will encrypt the password on the server.<br>Passwords must be at least five characters in length. |

**Example**

```
AssignClientPassword ('Brian', 'flyfisher');
```

This example assigns the password 'flyfisher' to the client named Brian.

## AssociateCAMIDToGroup

AssociateCAMIDToGroup creates an association between a TM1 user group and a CAMID.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
AssociateCAMIDToGroup(GroupName, CAMID, CAMIDDisplayValue);
```

| Argument | Description |
|---|---|
| GroupName | The name of the TM1 group you want to associate with the CAMID. |
| CAMID | The name of the CAMID group. If the CAMID does not exist, it will be created in the }ClientCAMAssociatedGroups control cube. |
| CAMIDDefDisplayValue | The alias of the CAMID group. |

## CellSecurityCubeCreate

CellSecurityCubeCreate creates a security cube from an existing cube using a reduced set of dimensions. This function, like all functions that update metadata, should not be used in the Data or Epilog tabs of a process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellSecurityCubeCreate (‘DataCube’, ‘0:0:1:0’);
```

| Argument | Description |
|---|---|
| Cube | Name of the data cube. |
| DimensionMap | String specifying whether the dimension at each position should be used in the security cube. The order of dimensions is the original cube order. A 1 for each included dimension and a 0 for an excluded one. Each value separated by a colon. |
| Boolean return | True if the operation succeeded. A major error otherwise. |
| Additional information | The GrantSecurityAccess property must be set for this TurboIntegrator process to succeed. Creates the cell security cube. |

#### Example

```
CellSecurityCubeCreate (‘DataCube’, ‘0:0:1:0’);
```

This example creates an RDCLS cube from the cube called Data Cube.

## CellSecurityCubeDestroy

CellSecurityCubeDestroy destroys a security cube that was created from an existing cube. This function, like all functions that update metadata, should not be used in the Data or Epilog tabs of a process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
CellSecurityCubeDestroy (‘DataCube’, ‘0:0:1:0’);
```

| Argument | Description |
|---|---|
| Cube | Name of the data cube. |
| Boolean return | True if the operation succeeded. A major error otherwise. |
| Additional information | The GrantSecurityAccess property must be set for this TurboIntegrator process to succeed. Destroys the cell security cube. |

**Example**

```
CellSecurityCubeDestroy (‘DataCube’);
```

## DeleteClient

DeleteClient deletes a client from the server. Changes applied through the DeleteClient function do not take effect until the Metadata procedure in a process is completed. This function, like all functions that update metadata, should not be used in the Data or Epilog tabs of a process

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DeleteClient(ClientName);
```

| Argument | Description |
|---|---|
| ClientName | The name of the client you want to delete from the server. |

**Example**

```
DeleteClient('Brian');
```

This example removes the client Brian from the server.

## DeleteGroup

DeleteGroup deletes a user group from the server. Changes applied through the DeleteGroup function do not take effect until the Metadata procedure in a process is completed. This function, like all functions that update metadata, should not be used in the Data or Epilog tabs of a process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
DeleteGroup(GroupName);
```

| Argument | Description |
|---|---|
| GroupName | The group you want to delete. |

**Example**

```
DeleteGroup('Finance');
```

This example deletes the Finance user group from the server.

## ElementSecurityGet

ElementSecurityGet retrieves the security level assigned to a specified group for a dimension element.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ElementSecurityGet(DimName, ElName, Group);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the element for which you are retrieving a security level. |
| ElName | The element for which you are retrieving a security level. |
| Group | The user group for which you are retrieving a security level. |

#### Example

```
ElementSecurityGet('Region', 'Germany', 'Budgeting');
```

This example returns the security level assigned to the Budgeting user group for the Germany element of the Region dimension.

## ElementSecurityPut

ElementSecurityPut assigns a security level to a specified group for a dimension element.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ElementSecurityPut(Level, DimName, ElName, Group);
```

| Argument | Description |
|---|---|
| Level | The security level you are assigning. There are six possible Level values:<br>• None<br>• Read<br>• Write<br>• Reserve<br>• Lock<br>• Admin |
| DimName | The parent dimension of the element for which you are assigning a security level. |
| ElName | The element for which you are assigning a security level. |
| Group | The user group for which you are assigning a security level. |

**Example**

```
ElementSecurityPut('Reserve', 'Region', 'Germany', 'Budgeting');
```

This example assigns Reserve security to the Budgeting group for the Germany element of the Region dimension.

## HierarchyElementSecurityGet

HierarchyElementSecurityGet retrieves the security level assigned to a specified group for a dimension element.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyElementSecurityGet(DimName, HierName, ElName, Group);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the element for which you are retrieving a security level. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | The element for which you are retrieving a security level. |
| Group | The user group for which you are retrieving a security level. |

**Example**

```
HierarchyElementSecurityGet('Region', 'Europe', 'Germany', 'Budgeting');
```

This example returns the security level assigned to the Budgeting user group for the Germany element. The element appears in the Europe hierarchy of the Region dimension.

## HierarchyElementSecurityPut

HierarchyElementSecurityPut assigns a security level to a specified group for a dimension element.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchyElementSecurityPut(Level, DimName, HierName, ElName, Group);
```

| Argument | Description |
|---|---|
| Level | The security level you are assigning. There are six possible Level values:<br>• None<br>• Read<br>• Write<br>• Reserve<br>• Lock<br>• Admin |
| DimName | The parent dimension of the element for which you are assigning a security level. |
| HierName | The name of the hierarchy within the dimension. |
| ElName | The element for which you are assigning a security level. |
| Group | The user group for which you are assigning a security level. |

**Example**

```
HierarchyElementSecurityPut('Reserve', 'Region', 'Europe', 'Germany', 'Budgeting');
```

This example assigns Reserve security to the Budgeting group for the Germany element. The element appears in the Europe hierarchy of the Region dimension.

## RemoveCAMIDAssociation

RemoveCAMIDAssociation removes all associations between TM1 user groups and a specified CAMID.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

## Syntax

```
RemoveCAMIDAssociation(CAMID, RemoveCAMID);
```

| Argument | Description |
|---|---|
| CAMID | The name of the CAMID group for which you want to remove all security associations. |
| RemoveCAMID | Determines if the specified CAMID is deleted from the }ClientCAMAssociatedGroups control cube.<br>0 leaves the CAMID in the }ClientCAMAssociatedGroups control cube.<br>1 deletes the CAMID from the }ClientCAMAssociatedGroups control cube. |

## RemoveCAMIDAssociationFromGroup

RemoveCAMIDAssociationFromGroup removes an association between a TM1 user group and a CAMID.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
RemoveCAMIDAssociationFromGroup(GroupName, CAMID);
```

| Argument | Description |
|---|---|
| GroupName | The name of the TM1 user group for which you want to remove the association. |
| CAMID | The name of the CAMID group for which you want to remove the association. |

## RemoveClientFromGroup

RemoveClientFromGroup removes a specified client from a user group.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
RemoveClientFromGroup(ClientName, GroupName);
```

| Argument | Description |
|---|---|
| ClientName | The client you want to remove. |
| GroupName | The user group from which you want to remove the client. |

#### Example

```
RemoveClientFromGroup('Brian', 'Finance');
```

This example removes the client Brian from the Finance user group.

## SetHierarchyGroupsSecurity

SetHierarchyGroupsSecurity sets the security level for all existing groups for the specified dimension hierarchy.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SetHierarchyGroupsSecurity(securityLevel, dimension, hierarchy)
```

| Argument | Description |
|---|---|
| securityLevel | The security level that you are assigning. There are six possible values:<br>• None<br>• Read<br>• Write<br>• Reserve<br>• Lock<br>• Admin |
| dimension | Name of the dimension. |
| hierarchy | Name of the dimension hierarchy. |

**Example**

```
SetHierarchyGroupsSecurity('Reserve', 'Region', 'Europe');
```

This example assigns Reserve security to all existing groups in the Europe hierarchy of the Region dimension.

## SetHierarchyElementGroupsSecurity

SetHierarchyElementGroupsSecurity sets the security level for a specified element from a hierarchy in a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SetHierarchyElementGroupsSecurity(securityLevel, dimension, hierarchy, element)
```

| Argument | Description |
|---|---|
| securityLevel | The security level you are assigning. There are six possible values:<br>• None<br>• Read<br>• Write<br>• Reserve<br>• Lock<br>• Admin |
| dimension | Name of the dimension. |
| hierarchy | Name of the dimension hierarchy. |
| element | The element for which you are assigning a security level. |

**Example**

```
SetHierarchyElementGroupsSecurity('Reserve', 'Region', 'Europe', 'Germany');
```

This example assigns Reserve security to the Germany element of the Europe hierarchy in the Region dimension.

## SetDimensionGroupsSecurity

SetDimensionGroupsSecurity sets the security level for all existing groups for the specified dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SetDimensionGroupsSecurity(securityLevel, dimension)
```

| Argument | Description |
|---|---|
| securityLevel | The security level you are assigning. There are six possible values:<br>• None<br>• Read<br>• Write<br>• Reserve<br>• Lock<br>• Admin |
| dimension | Name of the dimension. |

**Example**

```
SetDimensionGroupsSecurity('Reserve', 'Region');
```

This example assigns Reserve security to all existing groups in the Region dimension.

## SetElementGroupsSecurity

SetElementGroupsSecurity sets the security level for a specified element in a dimension.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SetElementGroupsSecurity(securityLevel, dimension, element)
```

| Argument | Description |
|---|---|
| securityLevel | The security level you are assigning. There are six possible values:<br>• None<br>• Read<br>• Write<br>• Reserve<br>• Lock<br>• Admin |
| dimension | Name of the dimension. |
| element | The element for which you are assigning a security level. |

**Example**

```
SetElementGroupsSecurity('Reserve', 'Region', 'Germany');
```

This example assigns Reserve security to the Germany element of the Region dimension.

## SecurityOverlayGlobalLockCell

SecurityOverlayGlobalLockCell is used to restrict the access rights of a node to read-only by locking it. It uses the global overlay so all users are affected. The overlay cube must be created prior to using this command. The elements provided in the address must be only for the dimensions used in the overlay.

The process must be configured to modify security data to successfully execute `SecurityOverlayGlobalLockCell`.

The function returns True if successful and a major error if unsuccessful.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SecurityOverlayGlobalLockCell(bLock, Cube, element1,..., elementN)
```

| Argument | Description |
|---|---|
| bLock | If 1 lock it. 0 unlock it |
| Cube | Name of the cube. |
| elementN | Overlay element name that defines the tuple. The order must match the original dimension order of the cube. |

**Example**

```
SecurityOverlayGlobalLockCell(1,’Sales’,’MA’);
SecurityOverlayGlobalLockCell(0,’Products’,’MA','2011’);
```

In the first example, there is only one dimension used for the overlay. The second example uses two dimensions.

## SecurityOverlayCreateGlobalDefault

SecurityOverlayCreateGlobalDefault is used to create or destroy a Security Overlay cube, and to set the overlay for a given area of a data cube.

Creating a data cube with a name that signifies an overlay cube will cause the data cube to be made into an overlay if the server is restarted. When the cube is loaded it will be configured as an overlay if a matching data cube is found.

Global overlays apply to all users.

The process must be configured to modify security data to successfully execute `SecurityOverlayCreateGlobalDefault`.

The function returns True if successful and a major error if unsuccessful.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SecurityOverlayCreateGlobalDefault (Cube, DimensionMap)
```

| Argument | Description |
|---|---|
| Cube | Name of the cube. |
| DimensionMap | String specifying whether the dimension at each position should be used in the overlay. The order of dimensions is the original cube order. A 1 for each included dimension and a 0 for an excluded one. Each value separated by a colon. |

#### Example

```
SecurityOverlayCreateGlobalDefault('DataCube',
        '0:0:1:0');
```

## SecurityOverlayDestroyGlobalDefault

SecurityOverlayDestroyGlobalDefault is used to destroy a Security Overlay cube, and to set the overlay for a given area of a data cube.

Creating a data cube with a name that signifies an overlay cube will cause the data cube to be made into an overlay if the server is restarted. When the cube is loaded it will be configured as an overlay if a matching data cube is found

Global overlays apply to all users.

The process must be configured to modify security data to successfully execute `SecurityOverlayDestroyGlobalDefault`.

The function returns True if successful and a major error if unsuccessful.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SecurityOverlayDestroyGlobalDefault (Cube)
```

| Argument | Description |
|---|---|
| Cube | Name of the cube. |

**Example**

```
SecurityOverlayDestroyGlobalDefault('DataCube');
```

## SecurityOverlayGlobalLockNode

SecurityOverlayGlobalLockNode is used to restrict the access rights of a node to read-only by locking it. It uses the global overlay so all users are affected. The overlay cube must be created prior to using this command. The elements provided in the address must be only for the dimensions used in the overlay.

The process must be configured to modify security data to successfully execute `SecurityOverlayGlobalLockNode`.

The function returns True if successful and a major error if unsuccessful.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SecurityOverlayGlobalLockNode(bLock, Cube, Address, [AddressDelimiter])
```

| Argument | Description |
|---|---|
| bLock | If 1 lock it. 0 unlock it |
| Cube | Name of the cube. |
| Address | Tokenized string sequence of overlay element names that define the tuple. The order must match the original dimension order of the cube. |
| AddressDelimiter | Optional character string used to separate element names in the Address parameter. Default value '\|'. |

**Examples**

```
SecurityOverlayGlobalLockNode(1,'Sales','MA');
SecurityOverlayGlobalLockNode(0,'Products','MA | 2011');
SecurityOverlayGlobalLockNode(0,'Products', 'MA : 2011', ':');
```

In the first example there is only one dimension used for the overlay. The other two examples use two dimensions.

## SecurityRefresh

SecurityRefresh reads all the security control cubes and regenerates the internal structures in the server that are used by TM1 API functions.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SecurityRefresh;
```

### Arguments

None.

## Server Manipulation TurboIntegrator Functions

These functions facilitate server manipulation.

The server manipulation functions are not valid in processes on TM1 Database 12.

### BatchUpdateFinish

BatchUpdateFinish instructs the server to exit batch update mode.

This function is valid in processes only.

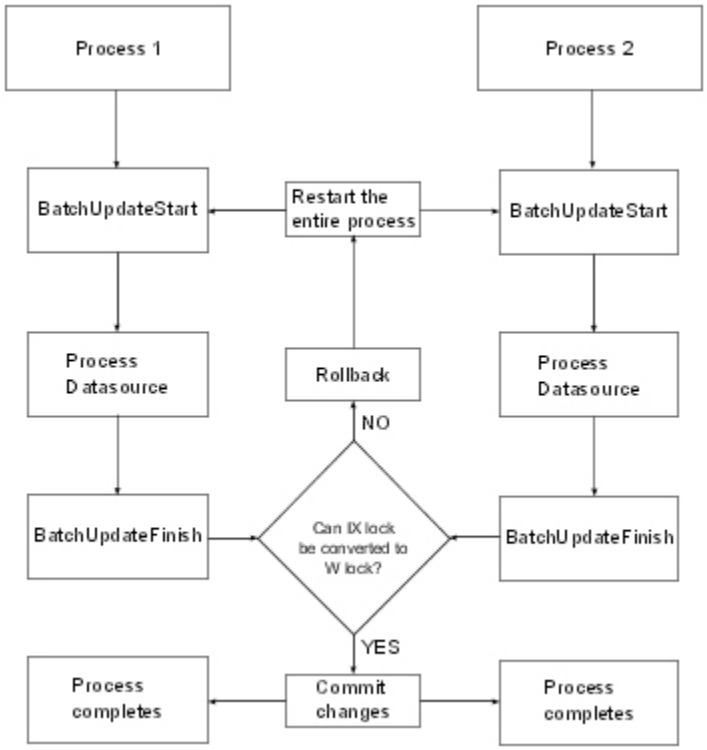
This function is not supported in processes on TM1 Database 12.

### Semantics

When multiple processes are running in batch update mode and applying changes to a single cube, the TM1 locking scheme may prevent one of the processes from updating the cube. This is by design; when one process obtains a lock to write changes to a cube, other processes will be prevented from writing to that cube in the interest of maintaining data integrity.

This locking scheme can be illustrated using an example of two processes, Process 1 and Process 2, that update a single cube.

- Both processes start and call the BatchUpdateStart function to initiate batch updates.
- Each process operates on a unique data source.
- Process 1 completes processing data and calls the BatchUpdateFinish function. The process obtains a write lock to the cube and commits changes.
- While Process 1 still holds a write lock to the cube, Process 2 completes processing data and calls the BatchUpdateFinish function. However, because Process 1 retains the lock, Process 2 cannot obtain a lock to the cube. All data changes applied in Process 2 are rolled back and Process 2 is restarted. This ensures data integrity.




Depending on the size of the datasource for Process 2, the data rollback and process re-execution can cause a noticeable decrease in performance. To address this performance issue, consider using the BatchUpdateFinishWait function in place of BatchUpdateFinish.

### Syntax

```
BatchUpdateFinish(SaveChanges);
```

| Argument | Description |
|---|---|
| SaveChanges | A flag that instructs the server to either save or discard changes committed while in batch update mode.<br>Specify 0 to save changes, 1 to discard changes. |

#### Example

```
BatchUpdateFinish(0);
```

This example instructs the server to save changes to TM1 data and exit batch update mode.

## BatchUpdateFinishWait

BatchUpdateFinishWait is identical to BatchUpdateFinish except the process waits until the lock becomes available and then commits changes. If a process calls BatchUpdateFinishWait but is unable to secure a cube write lock to commit changes, the process waits until the lock becomes available and then commits changes.

This function is valid in processes only.

Data changes applied in the process are not rolled back and the process is not re-executed.

**Note:** While waiting for the cube write lock, the process releases any read locks it acquired for other objects during process execution. Because these read locks are released before the process can commit changes to the cube, the objects for which the read locks are released can be modified *before* the cube is updated. This can lead to data inconsistency when using BatchUpdateFinishWait.

We recommend that BatchUpdateFinishWait be used only in controlled situations where you know that other processes are not modifying data or metadata related to the process that calls BatchUpdateFinishWait.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
BatchUpdateFinishWait(SaveChanges);
```

| Argument | Description |
|---|---|
| SaveChanges | A flag that instructs the server to either save or discard changes committed while in batch update mode. Specify 0 to save changes, 1 to discard changes. |

#### Example

```
BatchUpdateFinishWait(0);
```

This example instructs the server to save changes to TM1 data and exit batch update mode.

### *BatchUpdateStart*

BatchUpdateStart enables batch updates.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
BatchUpdateStart;
```

### Arguments

None.

## DisableBulkLoadMode

DisableBulkLoadMode disables bulk load processing.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

See “EnableBulkLoadMode” on page 1191 for details.

## EnableBulkLoadMode

EnableBulkLoadMode enables bulk load processing for a TurboIntegrator process.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

You can enable Bulk Load Mode in either the Prolog or Epilog section of a TurboIntegrator process. For efficiency, enable Bulk Load Mode in the first, or very close to the first, statement in the Prolog section of your process.

After enabling Bulk Load Mode in a process, it can only be disabled on the last line in the Epilog section. If you attempt to disable Bulk Load Mode anywhere else in the process, the process will not compile.

If the mode is enabled in one TurboIntegrator process, it remains enabled until explicitly disabled or until the chore completes. This means you can enable the mode in a process within a chore and then run a series of TurboIntegrator processes before disabling it. You can also enter and exit Bulk Load Mode repeatedly, using the mode only for certain critical parts of a chore.

Use the following TurboIntegrator commands to enable and disable Bulk Load Mode in a TurboIntegrator process.

```
EnableBulkLoadMode();
```

Use the following TurboIntegrator function only on the last line in the Epilog section of your TI process when using Bulk Load Mode.

```
DisableBulkLoadMode();
```

## RefreshMdxHierarchy

RefreshMdxHierarchy updates the MDX hierarchies in a server without requiring you to restart the server.

Use this function after configuring or editing the custom named hierarchy levels for a dimension in the }HierarchyProperties control cube.

For details on using named levels with dimensions, see the related section in the *TM1 for Developers* documentation.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
RefreshMdxHierarchy(dimensionName, hierarchy)
```

| Argument | Description |
|---|---|
| dimensionName | Optional string parameter to specify a specific dimension to update.<br>Leave this parameter blank to update all dimensions. |
| hierarchy | The name of the hierarchy within the dimension. This is an optional parameter. |

#### Example

Update all dimensions:

```
RefreshMdxHierarchy('');
```

To update only the customers dimension:

```
RefreshMdxHierarchy('customers');
```

## SaveDataAll

SaveDataAll saves all TM1 data from server memory to disk and restarts the log file.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Using SaveDataAll in a Chore

SaveDataAll commits all changes a chore makes prior to calling the SaveDataAll function.

While a chore is running, it accumulates locks on the objects it accesses. The commit operation initiated by the SaveDataAll function temporarily releases all these locks. Once the commit is complete, SaveDataAll reacquires all the locks it had before so it can continue to access the objects it was working on.

There is a brief window during the commit operation where the locks are released and another user or TurboIntegrator process could delete objects the original chore was using. When the original chore attempts to reacquire the locks on those objects, the objects will not be available and the chore will cease processing. In this case, an error similar to the following is written to the Tm1s.log file:

```
844 WARN 2008-04-01 16:40:09,734 TM1.Server TM1ServerImpl::FileSave could
not reacquire lock on object with index 0x200002ca
```

### Lock contention and using SaveDataAll at the end of TurboIntegrator processes

Using SaveDataAll as last command in a TurboIntegrator process can increase lock contention in TM1 TurboIntegrator processes.

In IBM TM1 versions, SaveDataAll was often added to the end of a TurboIntegrator process that loads data with logging disabled. The SaveDataAll provided a way to write data from memory to disk directly after a successful import, so that the newly imported data would not be lost in case of a mishap, such as a server crash.

However, adding SaveDataAll as the last command can result in numerous TurboIntegrator import processes, each one with SaveDataAll as last command. This technique worked in TM1 Version 9.0 and older due to the previous lock model which used only the global write lock. At any given time in earlier versions only one write operation could take place. Therefore competing concurrent SaveDataAll operations never occurred from multiple concurrent write operations.

Version 9.1 and newer introduced a more granular lock-by-object model that enables concurrent write operations, if these write operations do not compete for the same resources. If they do compete for the same resources, a lock contention occurs forcing one of the processes to rollback. So now two TurboIntegrator import processes may run simultaneously if they do not share any objects, for example, if they import into two different cubes.

The TurboIntegrator function SaveDataAll relies on the transaction logfile tm1s.log and involves all objects within a data model. Therefore, two TurboIntegrator import processes, both using the function SaveDataAll, cannot run in parallel: one will be executed, the other one (and its TurboIntegrator process) will be forced to rollback. The same is true if the TurboIntegrator processes are part of chores: only one chore will proceed to execute the TurboIntegrator function SaveDataAll, the other chore will be forced to rollback.

A rollback is undesirable from a performance point of view, as it increases the total execution time of a TurboIntegrator process or chore. Competing concurrent SaveDataAll operations will always lead to a lock contention and to a rollback.

There are two possible solutions to avoid competing concurrent SaveDataAll operations:

- Do not use the TurboIntegrator function SaveDataAll. Instead enable Cube Logging for the import cubes.
- If enabling Cube Logging for the import cubes cannot be done for performance reasons, within the TM1 application there should be only one process calling the TurboIntegrator function SaveDataAll. Use a stand-alone, single, distinct chore to execute the SaveDataAll operation.

### Syntax

```
SaveDataAll;
```

### Arguments

None.

## ServerShutdown

ServerShutdown shuts down a server running as an application. ServerShutdown cannot be used to shut down a server running as a Windows service.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ServerShutDown(SaveData);
```

| Argument | Description |
|---|---|
| SaveData | A Boolean value that indicates whether the server should save changes to disk before shutting down.<br>If SaveData = 0, the server shuts down without saving changes.<br>If SaveData = 1, the server saves changes from memory to disk before shutting down. |

#### Example

```
ServerShutdown(1);
```

This example shuts down the server and saves data to disk.

# Subset Manipulation TurboIntegrator Functions

These functions facilitate subset manipulation.

## HierarchySubsetAliasGet

HierarchySubsetAliasGet returns the alias attribute for a subset within a hierarchy. This function was introduced in Planning Analytics 2.0.9.10/TM1 Server 11.8.9 and cannot be used in previous versions.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetAliasGet(DimName, HierName, SubName);
```

| *Table 22. HierarchySubsetAliasGet syntax* | |
|---|---|
| **Argument** | **Description** |
| DimName | The parent dimension of the subset for which you want to retrieve the alias. |
| HierName | The name of a hierarchy within the dimension. |
| SubName | The subset within the specified hierarchy for which you want to retrieve the alias. |

**Example**

```
HierarchySubsetAliasGet('Region', 'European', 'Northern Europe');
```

This example retrieves the alias for the Northern Europe subset of the European hierarchy in the Region dimension.

## HierarchySubsetAliasSet

HierarchySubsetAliasSet sets the alias attribute to be used in an hierarchy subset. HierarchySubsetAliasSet returns 1 if successful, 0 otherwise.

This function is valid in TM1 TurboIntegrator processes only.

## Syntax

```
HierarchySubsetAliasSet(DimName, HierName, SubName, AliasName);
```

| **Argument** | **Description** |
|---|---|
| DimName | The parent dimension of the subset for which you want to set the alias. |
| HierName | The name of the hierarchy within the dimension. |
| SubName | The subset for which you want to set the alias. |
| AliasName | The alias you want to use in the subset. |

## HierarchySubsetCreate

HierarchySubsetCreate creates an empty public subset of a specified hierarchy and dimension.

When the optional `AsTemporary` argument is set to 1, the subset is temporary and persists only for the duration of the TurboIntegrator process or chore in which the subset is created.

**Note:**

For TM1 Server version 11.2.0 and earlier, temporary subsets were visible and usable only by the process that created it and any of its child processes. Temporary subsets were not visible to the ancestor and sibling processes. You could create same-named subsets in sibling child processes with the same parent process.

For TM1 Server version 11.3.0 and later, these temporary subsets are visible to the ancestor and sibling processes. If a parent TurboIntegrator process A invokes two child TurboIntegrator processes A1 and A2, and the child TurboIntegrator process A1 creates a temporary subset S, the temporary subset S exists for the duration of the parent TurboIntegrator process A. You cannot create a temporary subset with the same name S in the sibling TurboIntegrator process A2 since the subset is visible and usable by siblings A1 and A2.

While a temporary subset exists, the temporary subset takes precedence over any same-named public subset. If another TurboIntegrator function references a subset that exists in both a temporary and permanent state, the function operates upon the temporary subset.

There is no locking associated with a temporary subset, as a temporary subset is never saved. This can result in improved performance, because there is no need for TurboIntegrator to wait for locks to be released before operating upon a temporary subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetCreate(DimName, HierName, SubName, [AsTemporary]);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset you are creating. |
| HierName | The name of the hierarchy within the dimension. |
| SubName | The name you want to assign to the subset. |
| AsTemporary | This is an optional argument that specifies whether the subset being created is temporary. 1 indicates a temporary subset, 0 indicates a permanent subset.<br>If this argument is omitted, the subset is permanent. |

#### Example

```
HierarchySubsetCreate('Region', 'European', 'Northern Europe', 1);
```

This example creates the temporary Northern Europe subset of the European hierarchy in the Region dimension. You can use SubsetElementInsert to add elements to the subset.

## HierarchySubsetDeleteAllElements

HierarchySubsetDeleteAllElements deletes all elements from a public subset of a dimension hierarchy.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetDeleteAllElements(DimName, HierName, SubsetName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset from which you want to delete elements. |
| HierName | The name of the hierarchy within the dimension. |
| SubsetName | The subset from which you want to delete elements. This must be a public subset. TurboIntegrator cannot access private objects. |

**Example**

```
HierarchySubsetDeleteAllElements('Region', 'European', 'Central Europe');
```

This example deletes all elements from the Central Europe subset of the European hierarchy in the Region dimension.

## HierarchySubsetDestroy

HierarchySubsetDestroy deletes a subset from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetDestroy(DimName, HierName, SubName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset you are deleting. |
| HierName | The name of the hierarchy within the dimension. |
| SubName | The name of the subset you want to delete. |

**Example**

```
HierarchySubsetDestroy('Region', 'European', 'Northern Europe');
```

This example deletes the Northern Europe subset of the European hierarchy in the Region dimension.

## HierarchySubsetElementExists

HierarchySubsetElementExists determines whether a specific element exists within a specific public subset on the server from which a TurboIntegrator process is executed. HierarchySubsetElementExists cannot be used to determine if an element exists in a private subset.

If the element exists in the specified subset, the function returns 1, otherwise it returns 0.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetElementExists(DimName, HierName, SubsetName, ElementName);
```

| Argument | Description |
|---|---|
| DimName | The dimension parent of the subset containing the element whose existence you want to confirm. |
| HierName | The name of the hierarchy in the specified dimension. |
| SubsetName | The public subset containing the element whose existence you want to confirm. |
| ElementName | The element whose existence you want to confirm. The ElementName argument only accepts the element name and not the alias. |

**Example**

```
HierarchySubsetElementExists('Region', 'Eastern', 'Europe', 'Italy');
```

This example determines if the Italy element exists in the Europe subset of the Eastern hierarchy from the Region dimension.

## HierarchySubsetElementDelete

HierarchySubsetElementDelete deletes an element from a subset of a dimension hierarchy.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetElementDelete(DimName, HierName, SubName, Index);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset from which you want to delete an element. |
| HierName | The name of the hierarchy within the dimension. |
| SubName | The subset from which you want to delete an element. |
| Index | The index number of the element you want to delete from the subset. |

**Example**

```
HierarchySubsetElementDelete('Region', 'European', 'Northern Europe', 3);
```

This example deletes the third element from the Northern Europe subset of the European hierarchy in the Region dimension.

## HierarchySubsetElementGetIndex

HierarchySubsetElementGetIndex retrieves the index of an element in a subset of a dimension hierarchy.

The function returns the index of the first occurrence of the specified element. If the element does not exist in the subset or cannot be found, then zero is returned. If the dimension or subset cannot be found or an out-of-range start index is specified, then an error is thrown and the TurboIntegrator function is stopped.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetElementGetIndex(DimName, HierName, SubsetName, ElementName, StartIndex);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset. |
| HierName | The name of the hierarchy within the dimension. |
| SubsetName | The subset that contains the element. |

| Argument | Description |
|---|---|
| ElementName | The element name to search for in the subset. The ElementName argument accepts both the element name and the alias. |
| StartIndex | The index number to begin searching from. The value must be between 1 and the size of the subset. |

**Example**

```
HierarchySubsetElementGetIndex('Region', 'Country', 'Europe', 'Italy', 3);
```

This example retrieves the index for Italy from the Europe subset of the Country hierarchy in the Region dimension. The search starts at index 3.

## HierarchySubsetElementInsert

HierarchySubsetElementInsert adds an element to an existing subset in a dimension hierarchy.

This function is valid in TM1 TurboIntegrator processes only.

## Syntax

```
HierarchySubsetElementInsert(DimName, HierName, SubName, ElName, Position);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset to which you want to add an element. |
| HierName | The name of the hierarchy within the dimension. |
| SubName | The name of the subset to which you are adding an element. |
| ElName | The name of the element you want to add to the subset. The element must exist in the TM1 database. |
| Position | A value that indicates the index position of the element within the subset. |

**Example**

```
HierarchySubsetElementInsert('Region', 'European', 'Northern Europe', 'Finland',3);
```

This example adds the element Finland to the Northern Europe subset of the European hierarchy in the Region dimension. Finland is the third element in the subset definition.

## HierarchySubsetExists

HierarchySubsetExists determines if a specific public subset exists on the server from which a TurboIntegrator process is executed. The function returns 1 if the subset exists on the server, otherwise it returns 0. Note that this function cannot be used to determine the existence of private subsets.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetExists(DimName, HierName, SubsetName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension that is the parent of the subset whose existence you want to confirm. |
| HierName | The name of the hierarchy within the dimension. |
| SubsetName | The name of the public subset whose existence you want to confirm |

#### Example

```
HierarchySubsetExists('Region', 'Industrialized, 'Northern Europe');
```

This example determines if the Northern Europe subset exists within the Industrialized hierarchy of the Region dimension.

## HierarchySubsetGetSize

HierarchySubsetGetSize returns the number of elements in a subset of a dimension hierarchy.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetGetSize(DimName, HierName, SubsetName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset for which you want to determine size. |
| HierName | The name of the hierarchy within the dimension. |
| SubsetName | The subset for which you want to determine size. |

#### Example

```
HierarchySubsetGetSize('Region', 'Eastern', 'EurAsia');
```

This function returns the number of elements in the EurAsia subset of the Eastern hierarchy in the Region dimension.

## HierarchySubsetGetElementName

HierarchySubsetGetElementName returns the name of the element at a specified index location within a given subset of a dimension hierarchy.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetGetElementName(DimName, HierName, SubsetName, ElementIndex);
```

| Argument | Description |
|---|---|
| DimName | The parent of the subset from which you want to retrieve an element name. |
| HierName | The name of the hierarchy within the dimension. |
| SubsetName | The subset from which you want to retrieve an element name. |
| ElementIndex | A number representing the position within the subset of the element you want to retrieve. |

**Example**

```
HierarchySubsetGetElementName('Region', 'Western', 'Americas', 4);
```

This example returns the name of the fourth element in the Americas subset of the Western hierarchy in Region dimension.

## HierarchySubsetIsAllSet

HierarchySubsetIsAllSet sets a subset to use all elements of the parent dimension. HierarchySubsetIsAllSet returns 1 if successful, 0 otherwise.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetIsAllSet(DimName, HierName, SubName, Flag);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset for which you want to use all elements. |
| HierName | The name of the hierarchy within the dimension. |
| SubName | The subset for which you want to use all dimension elements. |
| Flag | Any non-zero value specifies that the subset uses all the current elements from the parent dimension and will dynamically update to use all elements from the parent dimension whenever the subset is called.<br>Specifying a zero value freezes the elements in the subset as the current set of all elements in the parent dimension. The subset will not dynamically update to use all dimension elements in the future. |

## HierarchySubsetMDXGet

HierarchySubsetMDXGet retrieves the MDX expression used to create a subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetMDXGet(DimName, HierName, SubName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset. |
| HierName | The name of the hierarchy within the dimension. |
| SubName | The subset for which you want to retrieve the MDX expression. |

#### Example

```
mdxString = HierarchySubsetMDXGet('Cities', 'Italy', 'testsubset');
```

## HierarchySubsetMDXSet

HierarchySubsetMDXSet applies a specified MDX expression to an existing public subset of a hierarchy.

If the passed MDX expression is valid, the specified subset is saved as a dynamic subset defined by the MDX expression.

If the passed MDX expression is an empty string, the subset is converted to a static subset that contains the elements that are in place when HierarchySubsetMDXSet is executed.

The function returns the number of elements that the subset contains.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
HierarchySubsetMDXSet(DimName, HierName, SubName, MDX_expression);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset. |
| HierName | The name of the hierarchy within the dimension. |
| SubName | The subset to which you want to apply the MDX expression. SubName must be a public subset. If this subset does not exist, an error is logged. |
| MDX_expression | The MDX expression that you want to apply to the subset. If the MDX expression is invalid, TurboIntegrator processing stops, the subset is not modified, and an error is logged.<br>If the MDX_expression argument is an empty string, the subset is converted to a static subset. |

#### Example

```
HierarchySubsetMDXSet('Cities', 'World', 'Sub1', '{ [Cities].[Cities].[level000].members }');
```

This example updates the `Sub1` subset of the `World` hierarchy to a dynamic subset that contains the current leaf elements of the `Cities` dimension. When leaf elements are added or removed from the `Cities` dimension, the `mySub1` subset is dynamically updated to reflect the changes in the parent dimension.

## PublishSubset

PublishSubset publishes a named private subset on the server. This function was introduced in Planning Analytics 2.0.9.10/TM1 Server 11.8.9 and cannot be used in previous versions.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
PublishSubset(DimName, SubName, OverwriteExistingSubset);
```

*Table 23. PublishSubset arguments*

| Argument | Description |
|---|---|
| DimName | The parent dimension of the private subset you want to publish. |
| SubName | The name of the private subset to be published. |
| OverwriteExistingSubset | This Boolean argument (1 or 0) determines if any existing identically named public subset are overwritten when the private subset is published.<br>If OverwriteExistingSubset is true (1) , any existing identically named public subset is overwritten when the private subset is published.<br>If this argument is false (0), the public subset is not overwritten, the private subset is not published, and an error is written to the TurboIntegrator log file. |

#### Example

```
PublishSubset('Region', 'Northern Europe', 1);
```

This example publishes the private Northern Europe subset of the Region dimension. If a public subset named Northern Europe already exists for the Region dimension, the public subset is overwritten then the private subset is published.

## SubsetAliasGet

SubsetAliasGet returns the alias attribute for a subset. This function was introduced in Planning Analytics 2.0.9.10/TM1 Server 11.8.9 and cannot be used in previous versions.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetAliasGet( DimName, SubName);
```

*Table 24. SubsetAliasGet syntax*

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset for which you want to retrieve the alias. |

| *Table 24. SubsetAliasGet syntax (continued)* | |
|---|---|
| **Argument** | **Description** |
| SubName | The subset for which you want to retrieve the alias. |

**Example**

```
SubsetAliasGet('Region', 'Central Europe');
```

This example retrieves the alias for the Central Europe subset of the Region dimension.

## SubsetAliasSet

SubsetAliasSet sets the alias attribute to be used in a subset. SubsetAliasSet returns 1 if successful, 0 otherwise.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetAliasSet( DimName, SubName, AliasName );
```

| **Argument** | **Description** |
|---|---|
| DimName | The parent dimension of the subset for which you want to set the alias. |
| SubName | The subset for which you want to set the alias. |
| AliasName | The alias you want to use in the subset. |

## SubsetCreate

SubsetCreate creates an empty public subset of a specified dimension.

When the `AsTemporary` argument is set to `1`, the subset is temporary and persists only for the duration of the TurboIntegrator process or a single-commit chore in which the subset is created. If a parent TurboIntegrator process invokes child TurboIntegrator processes by using the `ExecuteProcess` or `ExecuteProcessWithReturn` function, and the temporary subset is created in one of these child TurboIntegrator processes, the subset persists for the duration of the parent TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

**Note:**

For TM1 Server version 11.2.0 and earlier, temporary views and subsets were visible and usable only by the process that created it and any of its child processes. Temporary views and subsets were not visible to the ancestor and sibling processes. You could create same-named subsets in sibling child processes with the same parent process.

For TM1 Server version 11.3.0 and later, these temporary subsets are visible to the ancestor and sibling processes. If a parent TurboIntegrator process A invokes two child TurboIntegrator processes A1 and A2, and the child TurboIntegrator process A1 creates a temporary subset S, the temporary subset S exists for the duration of the parent TurboIntegrator process A. You cannot create a temporary subset with the same name S in the sibling TurboIntegrator process A2 since the subset is visible and usable by siblings A1 and A2.

A chore is a special case of a parent TurboIntegrator process that invokes a child TurboIntegrator process that is scheduled to run at a specific time. You can use two types of chores.

**Single-commit**
Within the scope / execution tree of a single-commit chore, a temporary subset of the same name can be created only for one child TurboIntegrator process.

**Multi-commit**
Within the scope / execution tree of a multi-commit chore, which commits after every child TurboIntegrator process, every child TurboIntegrator process can create a temporary subset of the same name because a temporary subset will not persist after a commit.

While a temporary subset exists, the temporary subset takes precedence over any same-named public or private subset. If another TurboIntegrator function references a subset that exists in both a temporary and permanent state, the function operates upon the temporary subset.

There is no locking associated with a temporary subset because a temporary subset is never saved. This can result in improved performance because there is no need for TurboIntegrator to wait for locks to be released before operating upon a temporary subset.

## Syntax

```
SubsetCreate(DimName, SubName, [AsTemporary]);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset you are creating. |
| SubName | The name that you want to assign to the subset. |
| AsTemporary | This is an optional argument that specifies whether the subset that is being created is temporary. 1 indicates a temporary subset. 0 indicates a permanent subset.<br>If this argument is omitted, the subset is permanent. |

**Example**

```
SubsetCreate('Region', 'Northern Europe', 1);
```

This example creates the temporary Northern Europe subset of the Region dimension. You can use `SubsetElementInsert` to add elements to the subset.

## SubsetCreateByMDX

SubsetCreateByMDX creates a public subset based on a passed MDX expression.

When the `AsTemporary` argument is set to 1, the subset is temporary and persists during the TurboIntegrator process or a single-commit chore in which the subset is created. If a parent TurboIntegrator process invokes child TurboIntegrator processes by using the `ExecuteProcess` or `ExecuteProcessWithReturn` function, and the temporary subset is created in one of these child TurboIntegrator processes, the subset persists during the parent TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

**Note:**

For TM1 Server version 11.2.0 and earlier, temporary views and subsets were visible and usable only by the process that created it and any of its child processes. Temporary views and subsets were not visible to the ancestor and sibling processes. You could create same-named subsets in sibling child processes with the same parent process.

For TM1 Server version 11.3.0 and later, these temporary subsets are visible to the ancestor and sibling processes. If a parent TurboIntegrator process A invokes two child TurboIntegrator processes A1 and A2, and the child TurboIntegrator process A1 creates a temporary subset S, the temporary subset S exists for the duration of the parent TurboIntegrator process A. You cannot create a temporary subset with the same name S in the sibling TurboIntegrator process A2 since the subset is visible and usable by siblings A1 and A2.

A chore is a special case of a parent TurboIntegrator process that invokes a child TurboIntegrator process. You can use two types of chores.

**Single-commit**
Within the scope / execution tree of a single-commit chore, a temporary subset of the same name can be created only for one child TurboIntegrator process.

**Multi-commit**
Within the scope / execution tree of a multi-commit chore, which commits after every child TurboIntegrator process, every child TurboIntegrator process can create a temporary subset of the same name because a temporary subset will not persist after a commit.

While a temporary subset exists, the temporary subset takes precedence over any same-named public or private subset. If another TurboIntegrator function references a subset that exists in both a temporary and permanent state, the function operates upon the temporary subset.

There is no locking associated with a temporary subset because a temporary subset is never saved. This can result in improved performance because there is no need for TurboIntegrator to wait for locks to be released before operating upon a temporary subset.

## Syntax

```
SubsetCreateByMDX(SubName, MDX_expression, DimName, 1)
```

| Argument | Description |
|---|---|
| SubName | The name that you want to assign to the subset. |
| MDX_expression | An MDX expression that returns a subset. |
| DimName | The dimension name. Specify the dimension name to avoid errors if the subset that is being created is empty. |
| AsTemporary | This is an optional argument that specifies whether the subset that is being created is temporary.<br>1 indicates a temporary subset. 0 indicates a permanent subset.<br>If this argument is omitted, the subset is permanent. |

### Example

This example creates a temporary subset based on an MDX expression that returns a subset that consists of all the dimensions whose names start with "plan_".

```
SubsetCreatebyMDX ( SubName , '{TM1FILTERBYPATTERN( {TM1SubsetALL( [ "| DimName |" ] )},
“plan_.*” )}' ,1);
```

This example returns an empty set as the MDX tries to create a subset that consists of Engine size of 2.0 models. Since only models with an Engine size of 1.6 or 1.8 exist, an empty set returns.

```
SubsetCreatebyMDX ( SubName , '{FILTER({TM1SUBSETALL([model])}, [model].[Engine Size] =
"2.0")}', 'model', 1);
```

#### Example of temporary subset used by parent process

In this example, two processes are created. The parent process, Process-A, calls the child process, Process-B. Process-B creates a temporary subset that Process-A uses as the data source.

Process-A/prolog:

```
ExecuteProcess('Process-B');
DatasourceDimensionSubset = 'My2003Months';
```

Process-B/prolog:

```
SubsetCreateByMdx('My2003Months', '{Descendants([plan_time].[plan_time].[2003], 3, LEAVES)}',
1);
```

## SubsetDeleteAllElements

SubsetDeleteAllElements deletes all elements from a public subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetDeleteAllElements(DimName, SubsetName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset from which you want to delete elements. |
| SubsetName | The subset from which you want to delete elements. This must be a public subset. TurboIntegrator cannot access private objects. |

#### Example

```
SubsetDeleteAllElements('Region', 'Central Europe');
```

This example deletes all elements from the Central Europe subset of the Region dimension.

## SubsetDestroy

SubsetDestroy deletes a subset from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetDestroy(DimName, SubName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset you are deleting. |
| SubName | The name of the subset you want to delete. |

#### Example

```
SubsetDestroy('Region', 'Northern Europe');
```

This example deletes the Northern Europe subset of the Region dimension.

## SubsetElementDelete

SubsetElementDelete deletes an element from a subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetElementDelete(DimName, SubName, Index);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset from which you want to delete an element. |
| SubName | The subset from which you want to delete an element. |
| Index | The index number of the element you want to delete from the subset. |

#### Example

```
SubsetElementDelete('Region', 'Northern Europe', 3);
```

This example deletes the third element from the Northern Europe subset of the Region dimension.

## SubsetElementExists

SubsetElementExists determines whether a specific element exists within a specific public subset on the server from which a TurboIntegrator process is executed. SubsetElementExists cannot be used to determine if an element exists in a private subset.

If the element exists in the specified subset, the function returns 1, otherwise it returns 0.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetElementExists(DimName, SubsetName, ElementName);
```

| Argument | Description |
|---|---|
| DimName | The dimension parent of the subset containing the element whose existence you want to confirm. |
| SubsetName | The public subset containing the element whose existence you want to confirm. |
| ElementName | The element whose existence you want to confirm. The ElementName argument only accepts the element name and not the alias. |

#### Example

```
SubsetElementExists('Region', 'Europe', 'Italy');
```

This example determines if the Italy element exists in the Europe subset of the Region dimension.

## SubsetElementGetIndex

SubsetElementGetIndex retrieves the index of an element in a subset. The function returns the index of the first occurrence of the specified element.

If the element does not exist in the subset or cannot be found, then zero is returned. If the dimension or subset cannot be found or an out-of-range start index is specified, then an error is thrown and the TurboIntegrator function is stopped.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetElementGetIndex(DimName, SubsetName, ElementName, StartIndex);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset. |
| SubsetName | The subset that contains the element. |
| ElementName | The element name to search for in the subset. The ElementName argument accepts both the element name and the alias. |
| StartIndex | The index number to begin searching from. The value must be between 1 and the size of the subset. |

#### Example

```
SubsetElementGetIndex('Region', 'Europe', 'Italy', 3);
```

This example retrieves the index for Italy from the Europe subset of the Region dimension. The search starts at index 3.

## SubsetElementInsert

SubsetElementInsert adds an element to an existing subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetElementInsert(DimName, SubName, ElName, Position);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset to which you want to add an element. |
| SubName | The name of the subset to which you are adding an element. |
| ElName | The name of the element you want to add to the subset. The element must exist in the TM1 database. |
| Position | A value that indicates the index position of the element within the subset. |

**Example**

```
SubsetElementInsert('Region', 'Northern Europe', 'Finland',3);
```

This example adds the element Finland to the Northern Europe subset of the Region dimension. Finland is the third element in the subset definition.

## SubsetExists

SubsetExists determines whether a specific public subset exists on the server from which a TurboIntegrator process is executed.

The function returns 1 if the subset exists on the server, otherwise it returns 0. Note that this function cannot be used to determine the existence of private subsets.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetExists(DimName, SubsetName);
```

| Argument | Description |
|---|---|
| DimName | The name of the dimension that is the parent of the subset whose existence you want to confirm. |
| SubsetName | The name of the public subset whose existence you want to confirm |

**Example**

```
SubsetExists('Region', 'Northern Europe');
```

This example determines if Northern Europe subset of the Region dimension exists on the server.

## SubsetExpandAboveSet

SubsetExpandAboveSet sets the Expand Above property for a subset. The function returns 1 if successful, otherwise it returns 0.

When this property is set to TRUE, children of a consolidation are displayed above the consolidation when the consolidation displays on a row, and to the left of the consolidation when the consolidation displays on a column.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetExpandAboveSet( DimName, SubsetName, ExpandAboveFlag);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset for which you want to set the Expand Above property. |
| SubsetName | The subset for which you want to set the Expand Above property. |

| Argument | Description |
|---|---|
| ExpandAboveFlag | Set ExpandAboveFlag to 1 to set the Expand Above property to TRUE. When this property is TRUE, consolidations expand above on rows and to the left on columns.<br>Set ExpandAboveFlag to 0 to set the Expand Above property to FALSE. When this property is FALSE, consolidations expand below on rows and to the right on columns. |

**Example**

```
SubsetExpandAboveSet('Region', 'Europe', 1 );
```

This example sets the Expand Above property to TRUE for the Europe subset of the Region dimension.

## SubsetFormatStyleSet

SubsetFormatStyleSet applies an existing display style to a named subset.

Display styles are defined for specific elements. If you apply an existing display style to a subset that includes elements that are not included in the display style, no formatting is applied to those elements.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetFormatStyleSet( DimName, SubsetName, FormatName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset to which you want to apply a display style. |
| SubsetName | The name of the subset to which you are applying a display style. |
| FormatName | The name of the existing display style you want to apply to the subset. |

**Example**

```
SubsetFormatStyleSet ('Region', 'Northern Europe', 'BoldCurrencyLeftJustified');
```

This example applies the BoldCurrencyLeftJustified display style to the Northern Europe subset of the Region dimension.

## SubsetGetElementName

SubsetGetElementName returns the name of the element at a specified index location within a given subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetGetElementName(DimName, SubsetName, ElementIndex);
```

| Argument | Description |
|---|---|
| DimName | The parent of the subset from which you want to retrieve an element name. |
| SubsetName | The subset from which you want to retrieve an element name. |
| ElementIndex | A number representing the position within the subset of the element you want to retrieve. |

**Example**

```
SubsetGetElementName('Region', 'Americas', 4);
```

This example returns the name of the fourth element in the Americas subset of the Region dimension.

## SubsetGetSize

SubsetGetSize returns the number of elements in a subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetGetSize(DimName, SubsetName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset for which you want to determine size. |
| SubsetName | The subset for which you want to determine size. |

**Example**

```
SubsetGetSize('Region', 'EurAsia');
```

This function returns the number of elements in the EurAsia subset of the Region dimension.

## SubsetIsAllSet

SubsetIsAllSet sets a subset to use all elements of the parent dimension. SubsetIsAllSet returns 1 if successful, 0 otherwise.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetIsAllSet(DimName, SubName, Flag);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset for which you want to use all elements. |
| SubName | The subset for which you want to use all dimension elements. |

| Argument | Description |
|---|---|
| Flag | Any non-zero value specifies that the subset uses all the current elements from the parent dimension and will dynamically update to use all elements from the parent dimension whenever the subset is called.<br><br>Specifying a zero value freezes the elements in the subset as the current set of all elements in the parent dimension. The subset will not dynamically update to use all dimension elements in the future. |

## SubsetMDXGet

SubsetMDXGet retrieves the MDX expression used to create a subset.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetMDXGet(DimName, SubName);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset. |
| SubName | The subset for which you want to retrieve the MDX expression. |

#### Example

```
mdxString = SubsetMDXGet('Cities', 'testsubset');
```

## SubsetMDXSet

SubsetMDXSet applies a specified MDX expression to a public or temporary subset.

If the passed MDX expression is valid, the specified subset is saved as a dynamic subset defined by the MDX expression.

If the passed MDX expression is an empty string, the subset is converted to a static subset that contains the elements that are in place when SubsetMDXSet is executed.

The function returns the number of elements that the subset contains.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
SubsetMDXSet(DimName, SubName, MDX_expression);
```

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset. |
| SubName | The subset to which you want to apply the MDX expression. SubName must be a public or temporary subset. If this subset does not exist, an error is logged. |

| Argument | Description |
|---|---|
| MDX_expression | The MDX expression that you want to apply to the subset. If the MDX expression is invalid, TurboIntegrator processing stops, the subset is not modified, and an error is logged.<br>If the MDX_expression argument is an empty string, the subset is converted to a static subset. |

**Examples**

```
SubsetMdxSet( 'YZProducts', 'mySub1', '{ Filter( [YZProducts].[YZProducts].members,
IsLeaf( [YZProducts].[YZProducts].currentmember ) ) }' );
```

This example updates the mySub1 subset to a dynamic subset that contains the current leaf elements of the YZProducts dimension. When leaf elements are added or removed from the YZProduct dimension, the mySub1 subset is dynamically updated to reflect the changes in the parent dimension.

One possible use of the SubsetMDXSet function is to apply an MDX expression to update an existing subset, and then immediately convert the subset to static.

```
SubsetMDXSet( 'YZProducts', 'mySub1', '{ [YZProducts].[YZProducts].[level000].members }' );
SubsetMDXSet( 'YZProducts', 'mySub1', '' );
```

This two-call sequence updates the mySub1 subset to a static subset that contains the current top-level elements of the YZProducts dimension.

The first call of SubsetMDXSet applies the { [YZProducts].[YZProducts].[level000].members } MDX expression to the mysub1 subset, resulting in a dynamic subset that includes all top-level (level 0) elements of the YZProducts dimension.

The second call of SubsetMDXSet passes an empty string as the MDX_expression argument, so the mysub1 subset is converted to a static subset.

# View Manipulation TurboIntegrator Functions

These functions pertain to view manipulation.

## PublishView

PublishView publishes a named private view on the server.

This function is valid in TurboIntegrator processes only.

### Syntax

```
PublishView(Cube, View, PublishPrivateSubsets, OverwriteExistingView);
```

| Argument | Description |
|---|---|
| Cube | The name of the cube containing the private view to be published. |
| View | The name of the private view to be published. |

| Argument | Description |
| --- | --- |
| PublishPrivateSubsets | This Boolean argument (1 or 0) determines if any private subsets present in the view should also be published.<br>If PublishPrivateSubsets is true (1), all private subsets used in the view are published along with the view.<br>If this argument is false (0), private subsets are not published. A public view cannot contain private subsets, so the view will not be published and an error will be written to the TurboIntegrator log file.<br>**Note:**<br>If a private subset contains another private subset as a user-defined consolidation, the subset can never be published using the PublishView function, regardless of the value of the PublishPrivateSubsets argument.<br>The PublishPrivateSubsets argument is ignored when the PublishView function is used with an MDX view. Any private subsets used in a MDX view remain private. The view fails to render for users without access to the private subset of the same name. |
| OverwriteExistingView | This Boolean argument (1 or 0) determines if any existing identically named public view should be overwritten when the private view is published.<br>If OverwriteExistingView is true (1) , any existing identically named public view will be overwritten when the private view is published.<br>If this argument is false (0), the public view will not be overwritten, the private view will not be published, and an error will be written to the TurboIntegrator log file. |

## DisableMTQViewConstruct

DisableMTQViewConstruct disables multi-threaded query processing when calculating a view to be used as a TurboIntegrator datasource for a single TurboIntegrator process. When MTQQuery=T in the tms1.cfg file, DisableMTQViewConstruct can be called to override this value on a TurboIntegrator process.

This function must appear in the Prolog, it has no effect in any other procedure within a process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

**Note:** If the value of the **MTQ** parameter is 1 (or OFF), this functionality is turned off entirely and cannot be overridden.

The value of **MTQQuery** can be overridden on a single TurboIntegrator process by calling the DisableMTQViewConstruct function.

If **MTQQuery**=T (the default), DisableMTQViewConstruct can be called to disable the functionality for individual TurboIntegrator processes.

After enabling EnableMTQViewConstruct in a process, it can only be disabled on the last line in the Epilog section. If you attempt to use DisableMTQViewConstruct anywhere else in the process, the process will not compile.

If the mode is enabled in one TurboIntegrator process, it remains enabled until explicitly disabled or until the chore completes. This means you can enable the mode in a process and then run a series of TurboIntegrator processes before disabling it.

#### Example

Use the following TurboIntegrator commands to disable multi-threaded query processing when calculating a view to be used as a TurboIntegrator datasource for a single TurboIntegrator process.

```
DisableMTQViewConstruct()
```

See also "EnableMTQViewConstruct" on page 1216.

## EnableMTQViewConstruct

EnableMTQViewConstruct enables multi-threaded query processing when calculating a view to be used as a TurboIntegrator datasource for a single TurboIntegrator process. When MTQQuery=F in the tms1.cfg file, EnableMTQViewConstruct can be called to override this value on a TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

**Note:** If the value of the **MTQ** parameter is 1 (or OFF), this functionality is turned off entirely and cannot be overridden.

The value of **MTQQuery** can be overridden on a single TurboIntegrator process by calling the EnableMTQViewConstruct function.

If **MTQQuery**=F in the tms1.cfg file on the server where this function is run, EnableMTQViewConstruct can be called to override this value on a single TurboIntegrator process.

You can enable EnableMTQViewConstruct in either the Prolog or Epilog section of a TurboIntegrator process. For efficiency, enable EnableMTQViewConstruct in the first, or very close to the first, statement in the Prolog section of your process.

#### Example

Use the following TurboIntegrator commands to enable multi-threaded query processing when calculating a view to be used as a TurboIntegrator datasource for a single TurboIntegrator process.

```
EnableMTQViewConstruct()
```

See also "DisableMTQViewConstruct" on page 1215.

After enabling EnableMTQViewConstruct in a process, it can only be disabled on the last line in the Epilog section. If you attempt to use disable DisableMTQViewConstruct anywhere else in the process, the process will not compile.

If the mode is enabled in one TurboIntegrator process, it remains enabled until explicitly disabled or until the chore completes. This means you can enable the mode in a process and then run a series of TurboIntegrator processes before disabling it.

## ViewColumnDimensionSet

ViewColumnDimensionSet sets a column dimension for a TM1 view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewColumnDimensionSet(CubeName, ViewName, DimName, StackPosition);
```

| Argument | Description |
|---|---|
| CubeName | The parent cube of the view for which you are setting the column dimension. |
| ViewName | The view for which you are setting the column dimension. |
| DimName | The dimension you want to set as a column dimension for the view. |
| StackPosition | A number that indicates the stack position of the dimension in the view. This is a 1-based number. 1 indicates the top-most stack position. 2 indicates a position below 1, and so on. |

#### Example

```
ViewColumnDimensionSet('98sales', 'Quarter1', 'Month',1);
```

This example sets Month as a column dimension for the 1Quarter view of the 98sales cube. In the event of stacked column dimensions, Month is placed in the top-most position.

## ViewColumnSuppressZeroesSet

ViewColumnSuppressZeroesSet suppresses or enables the display of columns containing only zero values in a TM1 cube view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewColumnSuppressZeroesSet(Cube, ViewName, Flag);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view for which you want to suppress or enable the display of zero values. |
| ViewName | The view for which you want to enable or suppress the display of zeroes. |

| Argument | Description |
|---|---|
| Flag | A binary value that enables or suppresses zeroes. Specify 1 to suppress the display of columns containing only zeroes in the view. Specify 0 to enable the display of columns containing only zeroes. |

**Example**

```
ViewColumnSuppressZeroesSet('99sales', '1st QuarterActuals', 1);
```

This example suppresses the display of any columns containing only zeroes in the 1st Quarter Actuals view of the 99sales cube.

## ViewConstruct

ViewConstruct constructs, pre-calculates, and stores a Stargate view in memory on a server. This function is useful for pre-calculating and storing large views so they can be quickly accessed after a data load or update.

This function is valid in processes only.

### Syntax

```
ViewConstruct(CubeName, ViewName);
```

| Argument | Description |
|---|---|
| CubeName | The cube from which you want to construct the view. |
| ViewName | The view you want to construct. This view must be an existing public native view on the server.<br>You cannot use an MDX view. An attempt to use an MDX view results in a `View not found` error. |

**Example**

```
Viewconstruct('99sales', '1st Quarter Actuals');
```

This example creates the view `1st Quarter Actuals` of the `99sales` cube.

## ViewCreate

ViewCreate creates an empty view of a specified cube.

When the optional `AsTemporary` argument is set to 1, the view is temporary and persists only for the duration of the TurboIntegrator process or chore in which the view is created.

**Note:**

For TM1 Server version 11.2.0 and earlier, temporary views were visible and usable only by the process that created it and any of its child processes. Temporary views were not visible to the ancestor and sibling processes. You could create same-named views in sibling child processes with the same parent process.

For TM1 Server version 11.3.0 and later, these temporary views are visible to the ancestor and sibling processes. If a parent TurboIntegrator process A invokes two child TurboIntegrator processes A1 and A2,

and the child TurboIntegrator process A1 creates a temporary view S, the temporary view S exists for the duration of the parent TurboIntegrator process A. You cannot create a temporary view with the same name S in the sibling TurboIntegrator process A2 since the view is visible and usable by siblings A1 and A2.

While a temporary view exists, the temporary view takes precedence over any same-named public view. If another TurboIntegrator function references a view that exists in both a temporary and permanent state, the function operates upon the temporary view.

Temporary objects have transaction scope. When a transaction is committed, all temporary objects are cleaned up. If a chore is run in single-commit mode where all processes in the chore are logically run within the context of one transaction, then temporary objects that are created in a process still exist, visible, and available for use, in subsequent processes run by the chore. However, in multi-commit mode, these processes are cleaned up at commit time of the transaction that wrapped the execution of the process that created the temporary object.

There is no locking associated with a temporary view, as a temporary view is never saved. This can result in improved performance, because there is no need for TurboIntegrator to wait for locks to be released before operating upon a temporary view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewCreate(Cube, ViewName, <AsTemporary>);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view you are creating. |
| ViewName | The name you want to assign to the view. |
| AsTemporary | This is an optional argument that specifies whether the view being created is temporary. 1 indicates a temporary view, 0 indicates a permanent view.<br>If this argument is omitted, the view is permanent. |

#### Example

This example creates a temporary view named 1st Quarter Actuals from the Sales cube.

```
ViewCreate('Sales', '1st Quarter Actuals', 1);
```

## ViewCreateByMDX

ViewCreateByMDX creates a view with a specified MDX expression.

When the optional `AsTemporary` argument is set to 1, the view is temporary and persists only for the duration of the TurboIntegrator process or chore in which the view is created.

**Note:**

For TM1 Server version 11.2.0 and earlier, temporary views were visible and usable only by the process that created it and any of its child processes. Temporary views were not visible to the ancestor and sibling processes. You could create same-named views in sibling child processes with the same parent process.

For TM1 Server version 11.3.0 and later, these temporary views are visible to the ancestor and sibling processes. If a parent TurboIntegrator process A invokes two child TurboIntegrator processes A1 and A2, and the child TurboIntegrator process A1 creates a temporary view S, the temporary view S exists for the duration of the parent TurboIntegrator process A. You cannot create a temporary view with the same

name S in the sibling TurboIntegrator process A2 since the view is visible and usable by siblings A1 and A2.

While a temporary view exists, the temporary view takes precedence over any same-named public view. If another TurboIntegrator function references a view that exists in both a temporary and permanent state, the function operates upon the temporary view.

Temporary objects have transaction scope. When a transaction is committed, all temporary objects are cleaned up. If a chore is run in single-commit mode where all processes in the chore are logically run within the context of one transaction, then temporary objects that are created in a process still exist, visible, and available for use, in subsequent processes run by the chore. However, in multi-commit mode, these processes are cleaned up at commit time of the transaction that wrapped the execution of the process that created the temporary object.

There is no locking associated with a temporary view, as a temporary view is never saved. This can result in improved performance because there is no need for TurboIntegrator to wait for locks to be released before operating upon a temporary view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewCreateByMDX(Cube, ViewName, MDX_expression , <AsTemporary>);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view you are creating. |
| ViewName | The name you want to assign to the view. |
| MDX_expression | A string value containing a valid MDX view expression. |
| AsTemporary | This is an optional argument that specifies whether the view being created is temporary. 1 indicates a temporary view; 0 indicates a permanent view.<br>If this argument is omitted, the view is permanent. |

#### Example

This example, based on the Planning Sample database, creates a temporary view named Account in the plan_BudgetPlan cube.

```
ViewCreateByMDX('plan_BudgetPlan', 'Account',
    'select {[plan_version].[FY 2003 Budget]} on 0,
    {[plan_business_unit].[10300]} on 1 from plan_budgetplan where
    [plan_department].[200][plan_chart_of_accounts].[41101][plan_exchange_rates].[local]
[plan_source].[goal][plan_time].[Jan-2003]'
    ,1);
```

## ViewDestroy

ViewDestroy deletes a view from the TM1 database.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewDestroy(Cube, ViewName);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view you are deleting. |
| ViewName | The name of the view you want to delete. |

**Example**

```
ViewDestroy('99sales', '1st Quarter Actuals');
```

This example deletes the 1st Quarter Actuals view of the 99sales cube.

## ViewExists

ViewExists determines whether a specific public view exists on the server from which a TurboIntegrator process is executed. The function returns 1 if the view exists on the server, otherwise it returns 0. Note that this function cannot be used to determine the existence of private views.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewExists(CubeName, ViewName);
```

| Argument | Description |
|---|---|
| CubeName | The name of the cube that is the parent of the view whose existence you want to confirm. |
| ViewName | The name of the public view whose existence you want to confirm |

**Example**

```
ViewExists('Inventory', 'FebClosing');
```

This example determines if FebClosing view of the Inventory cube exists on the server.

## ViewExtractFilterByTitlesSet

ViewExtractFilterByTitlesSet sets an option to filter by titles on consolidated values that are excluded from a view or any associated view extracts.

TM1 allows the storing of strings on calculated values. When you exclude a calculated value from a view or view extract you may want to exclude the message string also from the view.

**Note:** This function affects views as they exist on the server. The scope of this function is not restricted to extracts generated from a view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewExtractFilterByTitlesSet (Cube, ViewName, FilterByTitles, Temporary);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view for which you are setting the option. |
| ViewName | The view for which you are setting the option. |
| FilterByTitles | A binary value that turns the option on or off. Specify 0 to include titles stored on consolidated values. This is the current and default behavior. Specify 1 to exclude titles stored on consolidated values. |
| Temporary | A Boolean value that indicates whether the settings are temporary. |

**Example**

```
ViewExtractFilterByTitlesSet ('99sales', '1st QuarterActuals', 1, 0);
```

## ViewExtractSkipCalcsSet

ViewExtractSkipCalcsSet sets an option to include/exclude consolidated values in a view **and** any associated view extracts. A view extract is a view exported as an ASCII comma-delimited (.cma) file.

**Note:** This function affects views as they exist on the server. The scope of this function is not restricted to extracts generated from a view.

ViewExtractSkipCalcsSet is the equivalent of the Skip Consolidated Values option in the View Extract dialog box.

This function is valid only in Planning Analytics processes.

### Syntax

```
ViewExtractSkipCalcsSet (Cube, ViewName, Flag);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view for which you are setting the option. |
| ViewName | The view for which you are setting the option. |
| Flag | A binary value that turns the option on or off. Specify 1 to exclude consolidated values from the view extract. Specify 0 to include consolidated values.<br>The default is 1. |

**Example**

```
ViewExtractSkipCalcsSet ('99sales', '1st Quarter Actuals',1);
```

This example turns on the Skip Consolidated Values option for the 1st Quarter Actuals view. The view extract will not include any consolidated values.

### Note about the impact of enabling a specific combination of view manipulation functions

Consider the scenario when all of these conditions are true:

- the measure is a string
- ViewExtractSkipCalcsSet = 1
- ViewExtractSkipConsolidatedStringsSet = 0 (function is not used)
- ViewExtractSkipRuleValuesSet = 0 (function is not used)

In this scenario, the output is different, depending on whether you enable the ViewExtractSkipZeroesSet function.

- If you set ViewExtractSkipZeroesSet = 0, the Planning Analytics database enumerates every possible cube cell, not just the existing data cells. This situation is rather unusual, since enumerating all possible cells means that the number of cells scanned is the product of the sizes of all of the dimensions of the cube. This product can quickly become very large. In this mode, the ViewExtractSkipCalcsSet function skips all consolidated cells, even if the measure is a string.
- If you set ViewExtractSkipZeroesSet = 1, the Planning Analytics database scans only the cells actually in the cube. In this mode, a string stored on a consolidated cell is treated as a simple leaf (the cell after all has a simple value and is a leaf). Therefore, even though the ViewExtractSkipCalcsSet function is enabled, the entry is not skipped since this cell is not a calculated consolidated cell. In this case, if you want the entries to be skipped, you must enable the ViewExtractSkipConsolidatedStringsSet function.

## ViewExtractSkipConsolidatedStringsSet

ViewExtractSkipConsolidatedStringsSet sets an option to exclude strings on consolidated values that are excluded from a view or any associated view extracts. A view extract is a TM1 view exported as an ASCII comma-delimited (.cma) file.

TM1 allows the storing of strings on calculated values. When you exclude a calculated value from a view or view extract you may want to exclude the message string also from the view.

**Note:** This function affects views as they exist on the server. The scope of this function is not restricted to extracts generated from a view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewExtractSkipConsolidatedStringsSet (Cube, ViewName, Flag);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view for which you are setting the option. |
| ViewName | The view for which you are setting the option. |
| Flag | A binary value that turns the option on or off. Specify 0 to include strings stored on consolidated values. This is the current and default behavior. Specify 1 to exclude strings stored on consolidated values. |

**Note:** Read about the impact of enabling a specific combination of view manipulation functions.

**Example**

```
ViewExtractSkipConsolidatedStringsSet ('99sales', '1st QuarterActuals', 1);
```

This example turns on the Skip Rule for Consolidated String option for the extract created from the 1st Quarter Actuals view. The extract will not include any string on the consolidated value.

## ViewExtractSkipRuleValuesSet

ViewExtractSkipRuleValuesSet sets an option to include/exclude rule-calculated values in a view **and** any associated view extracts. A view extract is a TM1 view exported as an ASCII comma-delimited (.cma) file.

ViewExtractSkipRuleValuesSet is the equivalent of the Skip Rule Calculated Values option in the View Extract dialog box.

**Note:** This function affects views as they exist on the server. The scope of this function is not restricted to extracts generated from a view. Setting `ViewExtractSkipRuleValuesSet=1` causes a "rules off" mode, which turns off all rule evaluation. This means that not only are rule-calculated cells excluded, but their contribution to consolidations is also ignored. This can result in consolidated values in the view being different from the consolidated values in the cube.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewExtractSkipRuleValuesSet (Cube, ViewName, Flag);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view for which you are setting the option. |
| ViewName | The view for which you are setting the option. |
| Flag | A binary value that turns the option on or off. Specify 1 to exclude rule-calculated values from the extract. Specify 0 to include rule-calculated values. |

**Note:** Read about the impact of enabling a specific combination of view manipulation functions.

**Example**

```
ViewExtractSkipRuleValuesSet ('99sales', '1st QuarterActuals', 1);
```

This example turns on the Skip Rule Calculated Values option for the extract created from the 1st Quarter Actuals view. The extract will not include any rule-calculated values.

## ViewExtractSkipZeroesSet

ViewExtractSkipZeroesSet sets an option to include/exclude zero values in a view **and** any associated view extracts. A view extract is a TM1 view exported as an ASCII comma-delimited (.cma) file.

ViewExtractSkipZeroesSet is the equivalent of the Skip Zero/Blank Values option in the View Extract dialog box.

**Note:** This function affects views as they exist on the server. The scope of this function is not restricted to extracts generated from a view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewExtractSkipZeroesSet (Cube, ViewName, Flag);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view for which you are setting the Skip Zeroes option. |
| ViewName | The view for which you are setting the Skip Zeroes option. |
| Flag | A binary value that turns the option on or off. Specify 1 to exclude zeroes from the extract. Specify 0 to include zeros.<br>When UNDEFVALS is used to represent zeroes, the values are not excluded when the Flag argument is 1. |

**Note:** Read about the impact of enabling a specific combination of view manipulation functions.

#### Example

```
ViewExtractSkipZeroesSet ('99sales', '1st Quarter Actuals',1);
```

This example turns on the Skip Zeroes option for the extract created from the 1st Quarter Actuals view. The extract will not include any zero or blank values.

## ViewMDXSet

ViewMDXSet sets the MDX expression for an existing MDX view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewMDXSet(Cube, ViewName, MDX_expression);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view you are creating. |
| ViewName | The name you want to assign to the view. |
| MDX_expression | A string value containing a valid MDX view expression. |

#### Example

```
ViewMDXSet('Sales', 'Account',
    "select {[plan_version].[FY 2003 Budget]} on 0,
    {[plan_business_unit].[10300]} on 1 from plan_budgetplan where
    [plan_department].[200][plan_chart_of_accounts].[41101][plan_exchange_rates].[local]
[plan_source].[goal][plan_time].[Jan-2003]"
    );
```

This example sets the MDX expression for the "Account" view from the "Sales" cube.

## ViewMDXGet

ViewMDXGet retrieves the MDX expression for an existing MDX view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewMDXGet(Cube, ViewName);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view you are creating. |
| ViewName | The name you want to assign to the view. |

#### Example

```
ViewMDXGet('Sales', 'Account');
```

This example retrieves the MDX expression from the "Account" view.

## ViewRowDimensionSet

ViewRowDimensionSet sets a row dimension for a view.

This function is valid in TurboIntegrator processes only.

### Syntax

```
ViewRowDimensionSet(CubeName, ViewName, DimName, StackPosition);
```

| Argument | Description |
|---|---|
| CubeName | The parent cube of the view for which you are setting the row dimension. |
| ViewName | The view for which you are setting the row dimension. |
| DimName | The dimension you want to set as a row dimension for the view. |
| StackPosition | A number that indicates the stack position of the dimension in the view. This is a 1-based number. 1 indicates the left-most stack position. 2 indicates a position to the right of 1, and so on.<br>**Note:** It is possible for a TM1 client to set a Tm1p.ini parameter (BrowseDisplayReadsRightToLeft=T) that reverses the orientation of data in the Cube Viewer. When the orientation of data is reversed, the stack positions are also reversed. 1 indicates the right-most stack position. 2 indicates a position to the left of 1, and so on. |

**Example**

```
ViewRowDimensionSet('98sales', 'Quarter1', 'Month',1)
```

This example sets Month as a row dimension for the 1Quarter view of the 98sales cube. In the event of stacked row dimensions, Month is placed in the left-most position.

## ViewRowSuppressZeroesSet

ViewRowSuppressZeroesSet suppresses or enables the display of rows containing only zero values in a TM1 cube view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewRowSuppressZeroesSet(Cube, ViewName, Flag);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view for which you want to suppress or enable the display of zero values. |
| ViewName | The view for which you want to enable or suppress the display of zeroes. |
| Flag | A binary value that enables or suppresses zeroes. Specify 1 to suppress the display of rows containing only zeroes in the view. Specify 0 to enable the display of rows containing only zeroes. |

**Example**

```
ViewRowSuppressZeroesSet('99sales', '1st Quarter Actuals',1);
```

This example suppresses the display of any rows containing only zeroes in the 1st Quarter Actuals view of the 99sales cube.

## ViewSubsetAssign

ViewSubsetAssign assigns a named subset to a cube view.

**Note:** It is possible to create a temporary subset with the CreateSubset or CreateSubsetByMDX functions. If you attempt to use ViewSubsetAssign to assign a temporary subset to a permanent view, the function will fail with error notification.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewSubsetAssign(Cube, ViewName, DimName, SubName);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view to which you are assigning a subset. |
| ViewName | The view to which you are assigning a subset. |

| Argument | Description |
|---|---|
| DimName | The parent dimension of the subset you are assigning to the view. |
| SubName | The name of the subset you want to assign to the view. |

**Example**

```
ViewSubsetAssign('99sales', '1st Quarter Actuals', 'Month','Q1');
```

This example assigns the Q1 subset of the Month dimension to the 1st Quarter view.

## ViewSuppressZeroesSet

ViewSuppressZeroesSet suppresses or enables the display of all rows and columns containing only zero values in a TM1 cube view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewSuppressZeroesSet(Cube, ViewName, Flag);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view for which you want to suppress or enable the display of zero values. |
| ViewName | The view for which you want to enable or suppress the display of zeroes. |
| Flag | A binary value that enables or suppresses zeroes. Specify 1 to suppress the display of rows or columns containing only zeroes in the view. Specify 0 to enable the display of rows and columns containing only zeroes. |

**Example**

```
ViewSuppressZeroesSet('99sales', '1st Quarter Actuals',1);
```

This example suppresses the display of any rows or columns containing only zeroes in the 1st Quarter Actuals view of the 99sales cube.

## ViewTitleDimensionSet

ViewTitleDimensionSet sets a title dimension for a TM1 view.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewTitleDimensionSet(CubeName, ViewName, DimName);
```

| Argument | Description |
|---|---|
| CubeName | The parent cube of the view for which you are setting the title dimension. |
| ViewName | The view for which you are setting the title dimension. |
| DimName | The dimension you want to set as a title dimension for the view. |

**Example**

```
ViewTitleDimensionSet('98sales', 'Quarter1', 'Month');
```

This example sets Month as a title dimension for the 1Quarter view of the 98sales cube.

## ViewTitleElementSet

ViewTitleElementSet sets a title element for a TM1 view. ViewTitleElementSet is used in conjunction with the ViewTitleDimensionSet function.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
ViewTitleElementSet(CubeName, ViewName, DimName, Index);
```

| Argument | Description |
|---|---|
| CubeName | The parent cube of the view for which you are setting the title element. |
| ViewName | The view for which you are setting the title element. |
| DimName | The parent dimension of the title element. |
| Index | An index into the specified dimension that indicates the element to be set as the title element. |

**Example**

```
ViewTitleElementSet('98sales', 'Quarter1', 'Model',3);
```

This example sets the third element of the Model dimension as a title element for the Quarter1 view of the 98sales cube.

## ViewZeroOut

ViewZeroOut sets all data points in a view to zero.

This function is valid in TM1 TurboIntegrator processes only.

**Note:** When using ViewZeroOut on a cube which has UNDEFVALS enabled, the values in the view will be set to zero, not the UNDEFVAL state.

### Syntax

```
ViewZeroOut(Cube, ViewName);
```

| Argument | Description |
|---|---|
| Cube | The parent cube of the view you want to zero out. |
| ViewName | The view you want to zero out. |

#### Example

```
ViewZeroOut('99sales', '1st Quarter Actuals');
```

This example sets all data points in the 1st Quarter Actuals view to zero.

## Miscellaneous TurboIntegrator Functions

These functions facilitate miscellaneous tasks.

### AddInfoCubeRestriction

AddInfoCubeRestriction filters InfoCube data as it is pulled into TM1. Use this function to restrict the values that are imported for a specified characteristic. This function must be placed in the Prolog. The function can be called multiple times to filter more than one characteristic in a single process.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
AddInfoCubeRestriction(STRING CharactName, STRING sign,STRING compOperator,
STRING lowValue, STRING highValue)
```

| Argument | Description |
|---|---|
| STRING CharactName | Contains the technical name of the characteristic to be restricted. The data type has to be a character string with a length equal to or less than 30. |
| STRING sign | Contains either I (= inclusive) or E (= exclusive). Exclusive is the logical NOT for the restriction specified by this row. The data type has to be a character of length 1. |

| Argument | Description |
|---|---|
| STRING compOperator | Contains the relational comparative operator. The data type has to be a character string of length 2. Valid comparative operators are:<br>'EQ' = equal<br>'NE' = not equal<br>'LT' = less than<br>'GT' = grater than<br>'LE' = less or equal<br>'GE' = grater or equal<br>'BT' = between<br>'NB' = not between |
| STRING lowValue | Contains the low value for the operator specified in the row before. The data type has to be a character string with a length equal to or less than 60. |
| STRING highValue | Contains the high value for the operator specified two rows before. The data type has to be a character string with a length equal to or less than 60. It is only needed for the operators BT and NB, otherwise it is ignored, and in this case an empty string should be placed here. |

**Example**

The following example returns all characteristic values between 1997 and 2000.

```
AddInfoCubeRestriction('0CALYEAR','E','BT','1997','2000');
```

The following example returns all characteristic values not between 1997 and 2000.

```
AddInfoCubeRestriction('0CALYEAR','I','NB','1997', '2000') ;
```

The following example returns all characteristic values not equal to USD.

```
AddInfoCubeRestriction('0DOC_CURRCY', 'I', 'NE', 'USD','') ;
```

## ExecuteJavaN

ExecuteJavaN executes a Java™ TurboIntegrator process that returns a number. If you want to execute a Java TurboIntegrator process that returns a string, use ExecuteJavaS.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
ExecuteJavaN('JavaTIClass', ['OptionalParameter1', 'OptionalParameter2', ...] )
```

| Argument | Description |
|---|---|
| JavaTIClass | The fully qualified name of the Java TurboIntegrator class you want to execute. |
| OptionalParameters | Optional parameters that are passed to the Java TurboIntegrator process itself. You can pass as many parameters as necessary, including none.<br>You can pass only strings as parameters, you cannot pass numbers. You can use the StringToNumber TurboIntegrator function to pass numbers to Java TurboIntegrator scripts. For more information, see "StringToNumber" in the *TM1 Reference*. |

A Java TurboIntegrator class, which returns a number and can be called from ExecuteJavaN, must use the following pattern:

```
package com.example;

import com.ibm.cognos.tm1.javati.JavaTI;

@JavaTI
public class MyTestTI {
    public static double MyTestTI (String [] args) [
    ...
    return ...;
    }
}
```

### Example

```
ExecuteJavaN('com.example.MyTestTI', 'First', 'Second', 'Third');
```

## ExecuteJavaS

ExecuteJavaS executes a Java TurboIntegrator process that returns a string. If you want to execute a Java TurboIntegrator process that returns a number, use ExecuteJavaN.

This function is valid in processes only.

This function is not supported in processes on TM1 Database 12.

### Syntax

```
ExecuteJavaS('JavaTIClass', ['OptionalParameter1', 'OptionalParameter2', ...] )
```

| Argument | Description |
|---|---|
| JavaTIClass | The fully qualified name of the Java TurboIntegrator class you want to execute. |

| Argument | Description |
|---|---|
| OptionalParameters | Optional parameters that are passed to the Java TurboIntegrator process itself. You can pass as many parameters as necessary, including none.<br>You can pass only strings as parameters, you cannot pass numbers. You can use the StringToNumber TurboIntegrator function to pass numbers to Java TurboIntegrator scripts. For more information, see "StringToNumber" in the *TM1 Reference*. |

A Java TurboIntegrator class, which returns a string and is called from ExecuteJavaS, must use the following pattern.

```
package com.example;

import com.ibm.cognos.tm1.javati.JavaTI;

@JavaTI
public class MyTestTI {
    public static String MyTestTI (String [] args) [
    ...
    return ...;
    }
}
```

### Example

```
ExecuteJavaS('com.example.MyTestTI', 'First', 'Second', 'Third');
```

## Expand

Expand expands TurboIntegrator variable names, enclosed in % signs, to their values at run time. A common use of the Expand function is to pass the value of TurboIntegrator variables to the ODBCOutput function.

If the variable name represents a string variable, the entire variable expression must be enclosed on quotes. For example, "%V1%".

If Expand is fed with a numerical value, an implicit type conversion is performed and the numerical value is converted into a string.

That string has a fixed minimum length of 10 characters. If the converted number is too small to fill 10 characters, it is padded with leading spaces. Only three leading decimal characters are converted. For example, a numerical value of 0.123456789 is converted into the string "0.123".

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
Expand(String);
```

| Argument | Description |
|---|---|
| String | Any string that includes TurboIntegrator variable names enclosed in % signs. |

**Example**

```
ODBCOutPut( 'TransData', Expand( 'INSERT INTO SALES( MONTH, PRODUCT, SALES )
VALUES ( "%V0%", "%V1%",%V2% )' ) );
```

This example illustrates the use of the Expand function within the ODBCOutput function. The example inserts records into a relational table named Sales that consists of three columns: Month, Product, and Sales.

The Expand function converts the variables V0, V1, and V2 to their actual values within the view. Assuming that the first value in the view is 123.456, and is defined by the elements Jan and Widget

```
Expand( 'INSERT INTO SALES ( MONTH, PRODUCT, SALES ) VALUES ("%V0%", "%V1%",%V2% )' )
```

becomes

```
'INSERTINTO SALES ( MONTH, PRODUCT, SALES ) VALUES ( Jan, Widget,123.456 )'
```

at run time.

## FileExists

FileExists determines whether a specified file exists. The function returns 1 if the file exists, 0 if it does not.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
FileExists(File);
```

| Argument | Description |
|---|---|
| File | The name of a file. If a full parth is not specified, TM1 searches for the file in the server data directory. |

**Example**

```
FileExists('C:\tm1s7\pdata\model.dim');
```

This example determines if the model.dim file exists.

## LogOutput

LogOutput writes a message to the `tm1server.log` file and optionally the process log file when an error of a specified severity level is encountered in a TurboIntegrator process.

This function is valid in TM1 TurboIntegrator processes only.

### Prerequisite

To enable message logging from TurboIntegrator, you must add the `TM1.TILogOutput` debugger to the `tm1-log.properties` file and set the debugger to the wanted level. For example, adding `TM1.TILogOutput=DEBUG` to `tm1-log.properties` enables logging for all severity levels. For more information on the `tm1-log.properties` file, see "Configuring and Enabling Server Message Logging" in *TM1 Operations*.

### Syntax

```
LogOutput('SeverityLevel', 'MessageString', 'ProcessLog');
```

| Argument | Description |
|---|---|
| SeverityLevel | The severity level that initiates logging to the `tm1server.log` filelog files. Valid values for this argument are:<br>• `'DEBUG'`<br>• `'INFO'`<br>• `'WARN'`<br>• `'ERROR'`<br>• `'FATAL'` |
| MessageString | The message that you want to write to the `tm1server.log` filelog files. The message string can be a string that is enclosed in single quotation marks or can be another TurboIntegrator function that returns a string. |
| ProcessLog | Optional. If set to 1, this function also writes the message to the TurboIntegrator process log file in addition to the `tm1server.log` file.<br>If set to 0 or not defined, the function writes messages to just the `tm1server.log`. |

### Examples

```
LogOutput('WARN', 'TI process encountered a warning condition');
```

```
LogOutput('ERROR', TM1User(), 0);
```

```
LogOutput('INFO', 'TI process execution finished normally', 1);
```

## TM1User

TM1User returns a string giving the current TM1 client. When executed in a process that the user is running directly, it will return the user's TM1 client name. When executed in a chore that the user runs directly, it will also return the user's TM1 client name.

If run from a scheduled chore, it will return a name in the form *R*<chore name>*, for example, `R*UpdateRegionDimension`.

This function is valid in TM1 TurboIntegrator processes only.

### Syntax

```
TM1User()
```

## WildcardFileSearch

WildcardFileSearch lets you use wildcard characters to search for files in a specified directory.

The results of the WildCardFileSearch function may vary depending on the operating system in use. Files in a Windows directory are sorted in alphabetical order while files in a UNIX directory are sorted

in random order. Because the order of sorting varies between the operating systems, the identical WildCardFileSearch function executed against identical directories, one on Windows and one on UNIX, will yield different results.

This function is valid in TM1 TurboIntegrator processes only.

## Syntax

```
WildcardFileSearch( Pathname, PriorFilename);
```

| Argument | Description |
|---|---|
| Pathname | A pathname to files for which you want to search. The pathname must end in a filename, which can contain a wildcard sequence using the * and/or ? characters.<br>The ? wildcard character matches any single character.<br>The * wildcard character matches zero or more characters. |
| PriorFilename | The name of an existing file in the specified directory. This filename cannot contain wildcard characters. The wildcard search specified by the Pathname argument will commence *AFTER* this file.<br>If you pass an empty string as the PriorFilename argument, the WildcardFileSearch function returns the first file that matches the wildcard sequence specified by the Pathname argument. |

### Example

The following example shows the use of the WildcardFileSearch function to determine the first server log file generated in 2004:

```
file = WildcardFileSearch( 'C:\Program Files\Cognos\TM1\Custom\
TM1Data\SData\tm1s2004*.log', ' ');
```

This example returns the first file matching the wildcard sequence 'tm1s2004*.log' from the C:\Program Files\Cognos\TM1\Custom\TM1Data\SData\ directory.

Because server log files are named and saved with sequential time stamps, and because the second parameter to WildcardFileSearch is empty, the function returns the first server log file starting with the characters 'tm1s2004'. This would be the first server log file generated in the year 2004.

The following example shows the use of the WildcardFileSearch function to return the first server log file generated after tm1s20040211153827.log was generated:

```
file = WildcardFileSearch( 'C:\Program Files\Cognos\TM1\Custom\
TM1Data\SData\tm1s*.log', 'tm1s20040211153827.log
');
```

This example begins searching the C:\Program Files\Cognos\TM1\Custom\TM1Data\SData\ directory immediately after the tm1s20040211153827.log file, and returns the first subsequent file matching the 'tm1s*.log' wildcard sequence.

tm1s20040220175522.log is the first file that occurs after tm1s20040211153827.log and matches the wildcard sequence. Accordingly, the example returns tm1s20040220175522.log.

# TurboIntegrator Variables

IBM Planning Analytics variables are listed here by categories.

## TurboIntegrator Local Variables

When you execute a TurboIntegrator process, a set of implicit local variables is generated. Local variables exist only in the context of the process in which they are used, and are not available outside of the process. Local variables are destroyed when a process exits. These variables can be overwritten to manipulate a process. The following is the list of local variables:

### DatasourceASCIIDecimalSeparator

This TurboIntegrator local variable sets the decimal separator to be used in any conversion of a string to a number or a number to a string. If you set this variable you must also set the DatasourceASCIIThousandSeparator variable.

The character specified must be a standard ASCII printable character, with a decimal value between 33 and 127 inclusive.

#### Syntax

```
DatasourceASCIIDecimalSeparator='Char';
```

or

```
DatasourceASCIIDecimalSeparator=Char(xx);
```

| Argument | Description |
|---|---|
| Char | The ASCII character to be used as a separator.<br>The character can be specified as a character enclosed in single quotes, or as an ASCII Char decimal code without quotes. |

Either of the following examples sets the comma character (,) as the separator.

```
DatasourceASCIIDecimalSeparator=',';
```

```
DatasourceASCIIDecimalSeparator=Char(44);
```

### DatasourceASCIIDelimiter

This TurboIntegrator local variable sets the ASCII character to be used as a field delimiter when the DatasourceType is 'CHARACTERDELIMITED".

The character specified must be a standard ASCII printable character, with a decimal value between 33 and 127 inclusive.

#### Syntax

```
DatasourceASCIIDelimiter='Char';
```

or

```
DatasourceASCIIDelimiter=Char(xx);
```

| Argument | Description |
|---|---|
| Char | The ASCII character to be used as a delimiter.<br>The character can be specified as a character enclosed in single quotes, or as an ASCII Char decimal code without quotes. |

Either of the following examples sets the hyphen character (-) as the field delimiter.

```
DatasourceASCIIDelimiter='-';
```

```
DatasourceASCIIDelimiter=Char(45);
```

## DatasourceASCIIHeaderRecords

This TurboIntegrator local variable indicates the number of records to be skipped before processing the data source.

### Syntax

```
DatasourceASCIIHeaderRecords=N;
```

| Argument | Description |
|---|---|
| N | The number of records to be skipped before processing the data source. |

## DatasourceASCIIQuoteCharacter

This TurboIntegrator local variable sets the ASCII character used to enclose the fields of the source file when DatasourceType is 'CHARACTERDELIMITED'.

The character specified must be a standard ASCII printable character, with a decimal value between 33 and 127 inclusive.

### Syntax

```
DatasourceASCIIQuoteCharacter='Char';
```

or

```
DatasourceASCIIQuoteCharacter=Char(xx);
```

| Argument | Description |
|---|---|
| Char | The ASCII character that encloses fields in the data source.<br>The character can be specified as a character enclosed in single quotes, or as an ASCII Char decimal code without quotes. |

Either of the following examples sets the asterisk character (*) as the field delimiter.

```
DatasourceASCIIQuoteCharacter='*';
```

```
DatasourceASCIIQuoteCharacter=Char(42);
```

## DatasourceASCIIThousandSeparator

This TurboIntegrator local variable sets the thousands separator to be used in any conversion of a string to a number or a number to a string.

If you set this variable you must also set the DatasourceASCIIDecimalSeparator variable.

The character specified must be a standard ASCII printable character, with a decimal value between 33 and 127 inclusive.

### Syntax

```
DatasourceASCIIThousandSeparator='Char';
```

or

```
DatasourceASCIIThousandSeparator=Char(xx);
```

| Argument | Description |
|---|---|
| Char | The ASCII character to be used as a separator.<br>The character can be specified as a character enclosed in single quotes, or as an ASCII Char decimal code without quotes. |

Either of the following examples sets the period character (.) as the thousands separator.

```
DatasourceASCIIThousandSeparator='.';
```

```
DatasourceASCIIThousandSeparator=Char(46);
```

## DatasourceCubeview

This TurboIntegrator local variable sets the view to process if the DatasourceType is 'VIEW'.

### Syntax

```
DatasourceCubeview='ViewName';
```

| Argument | Description |
|---|---|
| ViewName | The name of the view to be processed. This must be an existing view of the cube specified by the DataSourceNameForServer variable. |

## DatasourceDimensionSubset

This TurboIntegrator local variable sets the subset to process if the DatasourceType is 'SUBSET.'

DatasourceNameForServer=*Dimension name* is also needed in conjunction with DATASOURCEDIMENSIONSUBSET so TM1 can identify where the subset is located.

### Syntax

```
DatasourceDimensionSubset='SubsetName';
```

| Argument | Description |
|---|---|
| SubsetName | The name of the subset to be processed. |

## DatasourceJsonRootPointer

This TurboIntegrator local variable stores a JSON-Pointer object, which identifies and locates a specific value in the JSON document.

This variable allows TurboIntegrator processes to use JSON files as data sources. Not specifying a root pointer results in the complete record (a JSON value) to be assigned to one variable.

### Syntax

```
DatasourceJsonRootPointer='RootPointerValue';
```

| Argument | Description |
|---|---|
| RootPointerValue | The JSON pointer, which identifies a specific value in the JSON document that is contained within the specified JSON resource. The JSON resource contains the record (or collection of records in a JSON array) that the TurboIntegrator process uses as a data source. |

## DatasourceJsonVariableMapping

This TurboIntegrator local variable maps specific values in the JSON document to individual variables.

This variable allows TurboIntegrator processes to use JSON files as data sources. The mapping is defined by using a JSON object in which each property (which identifies the variable by name) maps to a JSON-Pointer object that identifies the specific value in the JSON document that the TurboIntegrator processes uses as a data source.

### Syntax

```
DatasourceJsonVariableMapping='JsonStringValue';
```

| Argument | Description |
|---|---|
| JsonStringValue | A string that contains variables and values in JSON format. |

To build a JSON string with multiple lines, use multiple **JsonAdd()** functions.

For example, to create a JSON object that represents the following mapping:

```
{
    "vName": "/Name",
    "vStreet": "/Address/Street",
    "vCity": "/Address/City",
    "vSecondPhoneNumber": "/PhoneNumbers/1"
}
```

Use **JsonAdd()** functions to set the **DatasourceJsonVariableMapping** variable values:

```
DataSourceJsonVariableMapping = JsonAdd( '{}', 'vName', StringToJson( '/Name' ));
DataSourceJsonVariableMapping = JsonAdd( DataSourceJsonVariableMapping, 'vStreet',
StringToJson( '/Address/Street' ));
DataSourceJsonVariableMapping = JsonAdd( DataSourceJsonVariableMapping, 'vCity',
StringToJson( '/Address/City' ));
DataSourceJsonVariableMapping = JsonAdd( DataSourceJsonVariableMapping, 'vSecondPhoneNumber',
StringToJson( '/PhoneNumbers/1' ));
```

## DatasourceNameForServer

This TurboIntegrator local variable sets the name of the data source (.cma/.csv file, cube name, ODBC source) used by the server when executing the process.

### Syntax

```
DatasourceNameForServer='Name';
```

| Argument | Description |
|---|---|
| Name | For a .cma/.csv data source, the full path of the .cma file.<br>For cubes, the cube name prefaced with the string 'local:'.<br>For an ODBC source, the source name. |

## DatasourceNameForClient

This TurboIntegrator local variable sets the name of the data source (.cma file, cube name, ODBC source) used by the client when creating or editing the process.

### Syntax

```
DatasourceNameForClient='Name';
```

| Argument | Description |
|---|---|
| Name | For a .cma data source, the full path of the .cma file.<br>For cubes, the cube name prefaced with the string 'local:'.<br>For an ODBC source, the source name. |

## DatasourcePassword

This TurboIntegrator local variable sets the password used to connect to the data source.

### Syntax

```
DatasourcePassword='Password';
```

| Argument | Description |
|---|---|
| Password | The password used to connect to the data source set with DatasourceNameForServer. |

## DatasourceQuery

This TurboIntegrator local variable sets the query string to use with the data source.

### Syntax

```
DatasourceQuery='Query';
```

| Argument | Description |
|---|---|
| Query | The query string to use with the data source that was set with DatasourceNameForServer. |

## DatasourceType

This TurboIntegrator local variable sets the type of the data source.

### Syntax

```
DataSourceType='Type';
```

| Argument | Description |
|---|---|
| Type | Valid types include:<br>'CHARACTERDELIMITED', 'POSITIONDELIMITED', 'VIEW', 'SUBSET', 'ODBC' , 'OLEDBOLAP', 'NULL' |

## DatasourceUsername

This TurboIntegrator local variable sets the name used to connect to the data source.

### Syntax

```
DatasourceUserName='Name';
```

| Argument | Description |
|---|---|
| Name | The name used to connect to the data source set with DatasourceNameForServer. |

## MinorErrorLogMax

This TurboIntegrator local variable defines the number of minor errors that will be written to the TM1ProcessError.log file during process execution. If this variable is not defined in the process, the default number of minor errors written to the log file is 1000.

### Syntax

```
MinorErrorLogMax=N;
```

| Argument | Description |
|---|---|
| N | Value indicating the number of errors that should be written to the log file.<br>Specify an integer greater than zero to set the maximum number of errors written to the log file.<br>Specify a value of 0 to log no errors during process execution.<br>Specify a value of -1 to allow an unlimited number of minor errors to be written to the log file. |

```
The following table provides an example error log message
and the corresponding result.
```

| Example | Result |
|---|---|
| MinorErrorLogMax=750; | The log file will accept up to 750 errors. |
| MinorErrorLogMax=0; | No errors will be written to the log file. |
| MinorErrorLogMax=-1; | No limit on the number of errors written to the log file. |

## NValue

When the DatasourceType is 'VIEW', this TurboIntegrator local variable determines the value of the current cell when Value_Is_String is 0. (That is, when the current cell is numeric.)

### Syntax

```
Nvalue=N;
```

| Argument | Description |
|---|---|
| N | The value of the current cell. |

## OnMinorErrorDoItemSkip

This TurboIntegrator local variable instructs TurboIntegrator to skip to the next record when a minor error is encountered while processing a record.

This variable is useful in scenarios where a single bad field/value in a record causes multiple minor errors.

For example, if you have 100 CELLPUTN functions in a process and one of the fields in a given record is 'bad' or invalid, the minor error count is incremented by 100. (1 for each CELLPUTN function that encounters the error.) These 100 minor errors count towards the minor error limit defined by MinorErrorLogMax. A TurboIntegrator process fails when it surpasses the number of minor errors defined by MinorErrorLogMax.

If OnMinorErrorDoItemSkip=1; is included in the Prolog tab of the process, the process immediately skips to the next record when a 'bad' or invalid field is encountered in a source record. Using the previous example, this results in the minor error count being incremented by just 1, rather than 100.

### Syntax

```
OnMinorErrorDoItemSkip=N;
```

| Argument | Description |
|---|---|
| N | Value indicating if item should be skipped when a minor error is encountered.<br>1 (or any other non-zero value) dictates that the process should skip to the next record when a minor error is encountered.<br>0 indicates that TurboIntegrator should continue processing the current record when a minor error occurs. |

## SValue

When the DatasourceType is 'VIEW', this TurboIntegrator local variable determines the value of the current cell when Value_Is_String is not 0. (That is, when the current cell contains a string.)

### Syntax

```
Svalue='String';
```

| Argument | Description |
|---|---|
| String | The value of the current cell. |

## TM1ProcessError.log file

When a TurboIntegrator process encounters an error, it generates a TM1ProcessError.log file. This log file is saved to the data directory of the server on which the process resides.

**Note:** In a Planning Analytics on Cloud environment, the TM1ProcessError.log file is retained for three months. Any TM1ProcessError.log files that are older than three months are permanently deleted during the regularly scheduled maintenance window. If you want to retain your TM1ProcessError.log files beyond the three month maintenance interval, please compress them to a zip file. For more information on log file retention in Planning Analytics on Cloud, see Log file retention periods.

A TM1ProcessError.log file contains a list of errors that are encountered by the process. For each error encountered, the log file records the tab and line that caused the error, along with a brief description of the error.

When a process error log file is generated, TM1 assigns a unique name that lets you readily identify which TurboIntegrator process generated the error file and the time at which the file was created. File names are assigned with the following convention:

TM1ProcessError_<*time stamp*>_<*UID*>_<*process name*>.log.

In this convention:

- <*time stamp*> is the time (expressed as yyyymmddhhmmss GMT) at which the file was generated

- <*UID*> is a unique identifier expressed as MMMTTTTT. It is a combination of the millisecond (MMM) at which the error file was generated and the last five digits of the thread ID (TTTTT) of the process that caused the errors. If the thread ID contains less than five digits, the thread ID includes leading zeroes.
- <*process name*> is the name of the TurboIntegrator process that caused the errors

For example, an error file named TM1ProcessError_20220224203148_726008808_CreateSalesCube.log indicates that the error file was generated at 20:31:48:726 GMT on February 24, 2022 and that it contains errors that are caused by the CreateSalesCube process in thread 08808. Note the leading zero included in the thread ID.

Manual intervention is required to delete or archive these log files. A new log file is generated each time a TurboIntegrator process has an error (1 log file per TurboIntegrator execution).

Many TurboIntegrator process error logs might be generated for TurboIntegrator processes that run frequently and generate an error on each execution. Many log files generated in the TM1 database log directory might impact performance of the TM1 database when it creates or updates log files.

The Planning Analytics Administration ability to download log files might be impacted by a large number of files in the TM1 database log directory. It's recommended to limit the number of files in the TM1 database logging directory to under 2000 files.

### Value_Is_String

When the DatasourceType is 'VIEW', this TurboIntegrator local variable determines whether the current cell should be treated as a string or a numeric value.

#### Syntax

```
Value_Is_String=N;
```

| Argument | Description |
|---|---|
| N | Value indicating if the current cell is a string or a numeric value.<br>0 dictates that the cell is a number; anything else means the cell is treated as a string. |

## TurboIntegrator Global Variables

This type of TurboIntegrator variable is associated with an individual TM1 chore or with an individual process and any attendant sub-processes. There are two types of global variables: implicit and user-defined. Implicit global variables are described here. User-defined global variables are described in this document.

Global variables can be used in two ways:

- Global variables can be declared within a process that is part of a given chore. Once declared, the global variables are available to all other processes that are part of the chore. The variables persist while the chore is executing and for the duration of the current server session. Global variables are destroyed upon server shutdown.
- Global variables can be declared in one process and be made available to any subsequent processes called by the ExecuteProcess( ) function. These sub-processes must use the same global variable declaration statements (described in the following paragraphs) to access the global variables.

In the event that a global variable name is identical to a local variable name, the local variable definition takes precedence and overrides the global variable.

Global variables are declared in a TurboIntegrator process using one of the following two functions

### NumericGlobalVariable('VariableName');

Use this function to declare a numeric global variable.

### StringGlobalVariable('VariableName');

Use this function to define a string global variable.

## Implicit Global Variables

When you execute a TurboIntegrator process, a set of implicit global variables is generated. If the process generating the variables is part of a chore, these global variables are available to and can be shared by all other processes within the chore.

In addition, all implicit global variables in a process are available to and can be shared by any subsequent processes called by the ExecuteProcess( ) function.

Though implicit variables are generated by the TurboIntegrator process, you must declare a variable before it can be used in a process

Implicit global variables are declared in a TurboIntegrator process using the NumericGlobalVariable('VariableName');:

Click the following links for details on specific implicit global variables.

- DataMinorErrorCount.
- MetadataMinorErrorCount.
- ProcessReturnCode.
- PrologMinorErrorCount.

For example, to use the PrologMinorErrorCount implicit global variable in a process, you must first declare the variable as follows:

```
NumericGlobalVariable('PrologMinorErrorCount');
```

### DataMinorErrorCount

This TurboIntegrator global variable counts the minor errors that occur in the Data portion of a TurboIntegrator process. For each minor error encountered, the variable value is incremented by 1.

#### Syntax

```
DataMinorErrorCount=N;
```

| Argument | Description |
|---|---|
| N | The number of minor errors encountered in the Data portion of the process. |

### MetadataMinorErrorCount

This TurboIntegrator global variable counts the minor errors that occur in the Metadata portion of a TurboIntegrator process. For each minor error encountered, the variable value is incremented by 1.

#### Syntax

```
MetadataMinorErrorCount=N;
```

| *Table 25. MetadataMinorErrorCount arguments* | |
|---|---|
| **Argument** | **Description** |
| N | The number of minor errors encountered in the Metadata portion of the process. |

## ProcessReturnCode

This TurboIntegrator global variable stores the exit status of the most recently executed TurboIntegrator process.

### Syntax

```
ProcessReturnCode=StatusCode;
```

| **Status Code** | **Description** |
|---|---|
| ProcessExitByBreak() | Indicates that the process exited because it encountered a ProcessBreak function. |
| ProcessExitByChoreQuit() | Indicates that the process exited due to execution of the ChoreQuit function. |
| ProcessExitByChoreRollback() | Indicates that the process exited because it encountered a ChoreRollback function. |
| ProcessExitByProcessRollback() | Indicates that the process exited because it encountered a ProcessRollback function. |
| ProcessExitByQuit() | Indicates that the process exited because of an explicit `quit` command. |
| ProcessExitMinorError() | Indicates that the process executed successfully but encountered minor errors. |
| ProcessExitNormal() | Indicates that the process executed normally. |
| ProcessExitOnInit() | Indicates that the process aborted during initialization. |
| ProcessExitSeriousError() | Indicates that the process exited because of a serious error. |
| ProcessExitWithMessage() | Indicates that the process exited normally, with a message written to `tm1server.log`. |

## PrologMinorErrorCount

This TurboIntegrator global variable counts the minor errors that occur in the Prolog portion of a TurboIntegrator process. For each minor error encountered, the variable value is incremented by 1.

### Syntax

```
PrologMinorErrorCount=N;
```

| Argument | Description |
|---|---|
| N | The number of minor errors encountered in the Prolog. |

## TurboIntegrator User Variables

This type of variable is associated with an individual TM1 user, not with any particular process or chore. User variables can be manipulated from within any TurboIntegrator process or chore while the user with which the variable is associated is logged on to the server.

User variables must be explicitly declared. Once declared, user variables persist for the life of the user's TM1 session (until the user logs off or is disconnected from the server).

User variables are declared in a TurboIntegrator process using one of the following two functions:

- NumericGlobalVariable('VariableName');. Use this function to declare a numeric user variable.
- StringGlobalVariable('VariableName');. Use this function to define a string user variable.

User variables are created the first time such a declaration is encountered in any running TurboIntegrator process.

Once created, the variable name may be referenced and used just like any local or global variable, expect that the variable value persists across processes and chores only for as long as the user who created the variable is logged on to the server.

# MDX Function Support

All TM1 supported Microsoft-defined and TM1 specific functions are listed in this section.

## Support for Microsoft-defined MDX Expressions and Functions

TM1 supports Microsoft-defined MDX expressions and functions. The TM1 implementation of these functions and expressions is based on the definitions in the Microsoft MSDN library, which is available at the Microsoft MSDN website.

### Supported Member Expressions

The following MDX member expressions are supported.

- <dimension>.CURRENTMEMBER
- <member>.FIRSTCHILD
- <member>.FIRSTSIBLING
- <member>.LAG
- <member>.LASTCHILD
- <member>.LASTSIBLING
- <member>.LEAD
- <member>.NEXTMEMBER
- <member>.PARENT
- <member>.PREVMEMBER

### Supported Member Functions

The following MDX member functions are supported.

- ANCESTOR(...)
- COUSIN(...)

- OPENINGPERIOD(...)
- PARALLELPERIOD(...)

## Supported Numeric Functions

The following MDX numeric functions are supported.

- AGGREGATE(...)
- AVG(...)
- CORRELATION(...)
- COUNT(...)
- COVARIANCE(...)
- LINREGINTERCEPT(...)
- LINREGPOINT(...)
- LINREGR2(...)
- LINREGSLOPE(...)
- LINREGVARIANCE(...)
- MAX(...)
- MEDIAN(...)
- MIN(...)
- RANK(...)
- STDDEV(...)
- SUM(...)
- VAR(...)

## Supported Set Expressions

The following MDX set expressions are supported.

- <dimension>.MEMBERS
- <level>.MEMBERS
- <member>. CHILDREN
- <member>.SIBLINGS

## Supported Set Functions

The following MDX set functions are supported.

- ADDCALCULATEDMEMBERS(...)
- BOTTOMCOUNT(...)
- BOTTOMPERCENT(...)
- BOTTOMSUM(...)
- CROSSJOIN(...)
- DESCENDANTS(...)
- DISTINCT(...)
- DRILLDOWNLEVEL(...)
- DRILLDOWNLEVELBOTTOM(...)
- DRILLDOWNLEVELTOP(...)
- DRILLDOWNMEMBER(...)
- DRILLDOWNMEMBERBOTTOM(...)

- DRILLDOWNMEMBERTOP(...)
- DRILLUPMEMBER(...)
- DRILLUPLEVEL(...)
- EXCEPT(...)
- EXTRACT(...)
- FILTER(...)
- GENERATE(...)
- HEAD(...)
- HIERARCHIZE(...)
- INTERSECT(...)
- LASTPERIODS(...)
- ORDER(...)
- PERIODSTODATE(...)
- TOPCOUNT(...)
- TOGGLEDRILLSTATE(...)
- TOPPERCENT(...)
- TOPSUM(...)
- SUBSET(...)
- UNION(...)

### Supported Tuple Expressions

The following MDX tuple expressions are supported.

- <set>.CURRENTMEMBER **Note**: This expression is not supported in TM1 Database 12.
- <set>[.ITEM](...)

## TM1 specific MDX functions

TM1 supports several TM1 specific MDX expressions. You can apply these expressions while developing MDX applications to run against the server or when creating/editing dynamic subsets in the Expression Window of the Subset Editor.

### TM1DRILLDOWNMEMBER( <set1>, <set2>|ALL [,RECURSIVE] )

This TM1 specific MDX function is similar to the DRILLDOWNMEMBER function from Microsoft , but it has been adjusted to match the functionality of the Expand button {bmct expand_button.bmp} on the Subset Editor.

#### Syntax

ALL means drilldown all the members in <*set1*>.

RECURSIVE means that when one member from <*set1*> is being drilled down upon, every consolidated member resulting from that expansion will also be recursively drilled down until level 0 (TM1 level 0 ) is reached.

### TM1FILTERBYPATTERN( <set>, <pattern_str>, [<attribute>] )

This TM1 -specific MDX function returns all the members in a specified <*set*> with names matching the pattern <*pattern_str*>. You can also use an optional third parameter, <*attribute*>, to search for attribute values.

### Syntax

The syntax of <*pattern_str*> is the same used for the Select By Regular Expression option on the Set Editor.

#### Example

```
TM1FILTERBYPATTERN(TM1FILTERBYLEVEL(TM1SUBSETALL([Income Statement Account].
[Income Statement Account]) , 0),"0099*","MEMBER_NAME")
```

This example returns all MEMBER_NAME attributes for 0-level members from the Income Statement Account dimension that match the pattern string 0099*.

## TM1FILTERBYLEVEL( <set>, <level_number>)

This TM1 specific MDX function returns all the members in <*set*> of the specified <*level_number*>.

### Syntax

<*level_number*> is a number specifying the TM1 level number not an MDX level number.

## TM1Member

This function returns a member from a specified tuple.

A null member reference is returned when any of the following conditions are encountered:

- A null Tuple parameter
- An out-of-range numeric Index parameter
- A dimension or hierarchy parameter not found in the passed tuple.

### Syntax

```
TM1Member(Tuple, MemberSpecifier);
```

| Argument | Description |
|---|---|
| Tuple | An expression that resolves to a tuple. |
| MemberSpecifier | This parameter can be either a 0-based numeric index into the tuple or the name of a dimension/ hierarchy associated with the tuple. The following examples show both parameter types. |

#### Example

```
TM1Member ( [model].Members.Item(23) ,0 ) ]
```

This example uses a numeric index into the tuple as the MemberSpecifier argument.

```
TM1Member( [model].Members.Item(23), [Model] ) ]
```

This example uses the name of a dimension associated with the tuple as the MemberSpecifier argument.

## TM1SORT( <set>, ASC|DESC )

This TM1 specific MDX function sorts <*set*> alphabetically.

### Syntax

ASC sorts A-Z

DESC sorts Z-A

## TM1SORTBYINDEX( <set>, ASC|DESC )

This TM1 specific MDX function sorts *<set>* by the index value of the members.

### Syntax

ASC sorts by ascending index value.

DESC sorts by descending index value.

## TM1SUBSETALL([<dimname>])

This TM1 specific MDX function returns the TM1 subset All of *<dimname>*.

### Syntax

```
TM1SubsetAll([<dimname>]);
```

## TM1SubsetToSet

This function returns the members of a TM1 subset. TM1SubsetToSet is equivalent to the *<dimension>.<subsetname>* expression, but does not require string literals. Instead, TM1SubsetToSet lets you use expressions that resolve to the appropriate dimension and subset.

If a private and a public subset have identical names, enter the scope parameter to specify the scope of the search as "`private`" or "`public`".

### Syntax

```
TM1SubsetToSet(Dimension_exp, Subset_exp, Scope);
```

| Argument | Description |
|---|---|
| Dimension_exp | An expression that resolves to a valid TM1 dimension name.<br>Use [dimension] or [dimension].[hierarchy] to specify a dimension. |
| Subset_exp | An expression that resolves to a valid subset of the dimension returned by Dimension_exp.<br>When resolving an expression for a subset, the server searches first in the private subset list and then in the public list. |
| Scope (optional) | The subset list to search, which is specified as `private` or `public`.<br>Specify `public` to search the public list only.<br>If this parameter is not specified and a private and a public subset have the same name, the private subset is returned. |

**Example**

MDX sample code with dimension name `[Corp Planning Hry]` in the last line:

```
{INTERSECT(EXCEPT(DESCENDANTS([Corp Planning Hry].[Fixed Assets]),
TM1FILTERBYLEVEL(DESCENDANTS([Corp Planning Hry].[Fixed Assets]),0)),{
TM1SubsetToSet([Corp Planning Hry],"elist")}),[Corp Planning Hry].[FixedAssets]}
```

MDX sample code with fully qualified dimension name `[Corp Planning Hry].[Corp Planning Hry]` and `[Corp Planning Hry].[FixedAssets]` in the last line:

```
{INTERSECT(EXCEPT(DESCENDANTS([Corp Planning Hry].[Fixed Assets]),
TM1FILTERBYLEVEL(DESCENDANTS([Corp Planning Hry].[Fixed Assets]),0)),{
TM1SubsetToSet([Corp Planning Hry].[Corp Planning Hry],"elist")}),
[Corp Planning Hry].[FixedAssets]}
```

**Example**

MDX sample code with two subsets, one private, one public, both named 'MySub' under a `Markets` dimension. The following queries on columns from [QB] return the content of the private subset:

```
select TM1SubsetToSet( [QBMarkets].[QBMarkets], 'MySub' )
```

Or:

```
select TM1SubsetToSet( [QBMarkets].[QBMarkets], 'MySub', 'private' )
```

The following query on columns from [QB] returns the content of the public subset:

```
select TM1SubsetToSet( [QBMarkets].[QBMarkets], 'MySub', 'public' )
```

### TM1TupleSize

This function returns the number of members in a tuple.

### Syntax

```
TM1TupleSize(Tuple);
```

| Argument | Description |
|---|---|
| Tuple | An expression that resolves to a tuple.<br>The function returns 0 if the Tuple argument does not resolve to a valid tuple, or of the tuple is null or empty. |

## TM1 specific MDX expressions

TM1 supports several TM1 specific MDX expressions. You can apply these expressions while developing MDX applications to run against the server or when creating/editing dynamic subsets in the Expression Window of the Subset Editor.

### <dimension>.<subsetname>

This TM1 specific MDX expression returns members of <subsetname> in <dimension>.

### Syntax

Since the same syntax ( <dimension>.IDENTIFIER ) is used for members and levels, a subset with the same name of a member or a level will never be instantiated.

When searching for a subset, the server searches first in the private subset list and then in the public list.

## <member>.ANCESTORS

This TM1 specific MDX expression returns the ancestors of <member>.

### Syntax

For example, assuming the following hierarchy of the Month dimension:

- Year
  - 1 Quarter
    - Jan
    - Feb
    - Mar

The following expression:

```
month.month.jan.ANCESTORS
```

returns the set

```
{ 1Quarter, Year }.
```

**Note:** Where an element has more than one immediate parent, the expression returns the set containing the first parent in the default hierarchy. To see all the ancestors of an element, use ANCESTORSALL instead.

## <member>.WEIGHT

This TM1 specific MDX expression returns the weight of <member>.

### Syntax

The weighting property controls how a child member rolls up to its immediate parent, whether that child is also a parent of another consolidation or just a leaf member. Top level consolidations do not have weight.

# Data spread keyboard shortcuts

The following tables include details for all spreading methods that you can use in Planning Analytics Workspace.

You can specify the direction for data spreading by using these codes:

**Down**
|

**Up**
^

**Left**
<

**Right**
>

The default data action is Replace. To specify subtract, use a tilde (~). Use a plus sign (+) to specify the Add data action.

The following list summarizes the spreading methods that you can use:

**Proportional**

P

Example:

P<>100

Proportionally spreads the value 100 to all leaf cells on the row of insertion, and replaces the existing cell values.

**Equal spread**

S

Example:

S+|^200

Equally spreads the value 200 to all leaf cells on the column of insertion, and adds the product of spreading to the existing cell values.

**Repeat**

R

Example:

R~<50

Subtracts the value 50 from all leaf cells to the left of the insertion point.

**Clear**

C

Example:

C|^<>

Clears values from all leaf cells in the view.

**Percent change**

P%

Example:

P%+|^<>10

Applies a percent change of 10% to all leaf values, adds the product to the existing cell values, and increments all leaves by 10%.

**Repeat leaves**

LR

Example:

LR+*2100

Adds 2100 to all leaves of the consolidation. If you omit *, it copies 2100 only to populated leaf cells.

**Equal leaves**

LS

Example:

LS500

Distributes 500 equally across populated leaves of a consolidation.

**Straight line**

SL

Example:

SL>100:200

Replaces all leaf values to the right of the point of insertion, with a start value of 100 and an end value of 200.

Can be used across a single row or column, but not across rectangular ranges.

**Growth percentage**

GR

Example:

GR|300:25

Applies a 25% growth percentage to the starting value of 300 and replaces all leaf values below the point of insertion.

Can be used across a single row or column, but not across rectangular ranges.

## Proportional data spreading method

The Proportional spread method distributes a specified value among cells proportional to existing cell values.

For example, consider the following view in which the values for Argentina in the months Jan, Feb, and Mar are 10, 30, and 60, respectively.

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 100.00 | 10.00 | 30.00 | 60.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 |
| Brazil | 0.00 | 0.00 | 0.00 | 0.00 |
| Chile | 0.00 | 0.00 | 0.00 | 0.00 |
| Denmark | 0.00 | 0.00 | 0.00 | 0.00 |

The sum of these values is 100, with the value in Jan accounting for 10% of the sum, the value in Feb accounting for 30%, and the value in Mar accounting for 60%.

When you proportionally distribute the value 300 across these cells and select the Replace update action, the result is as follows.

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 300.00 | 30.00 | 90.00 | 180.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 |
| Brazil | 0.00 | 0.00 | 0.00 | 0.00 |
| Chile | 0.00 | 0.00 | 0.00 | 0.00 |
| Denmark | 0.00 | 0.00 | 0.00 | 0.00 |

These values are proportionally equivalent to the values that existed before you apply data spreading.

- Jan contains the value 30, which is 10% of 300
- Feb contains the value 90, which is 30% of 300
- Mar contains the value 180, which is 60% of 300

To distribute the value 300 proportionally across cells, equivalent to the values that existed before, type the following code:

```
P>300
```

## Equal data spreading method

The Equal Spread method distributes a specified value equally across the cells in a view.

For example, consider the following view where a range of 12 cells is selected.

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 165.00 | 25.00 | 75.00 | 65.00 |
| Belgium | 185.00 | 45.00 | 85.00 | 55.00 |
| Brazil | 165.00 | 35.00 | 55.00 | 75.00 |
| Chile | 145.00 | 35.00 | 65.00 | 45.00 |

When you equally spread the value 60 to these cells and select the Add update action, the value is equally spread across the range and added to the existing cell values. The result is that each cell value is incremented by 5 (60/12=5).

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 180.00 | 30.00 | 80.00 | 70.00 |
| Belgium | 200.00 | 50.00 | 90.00 | 60.00 |
| Brazil | 180.00 | 40.00 | 60.00 | 80.00 |
| Chile | 160.00 | 40.00 | 70.00 | 50.00 |

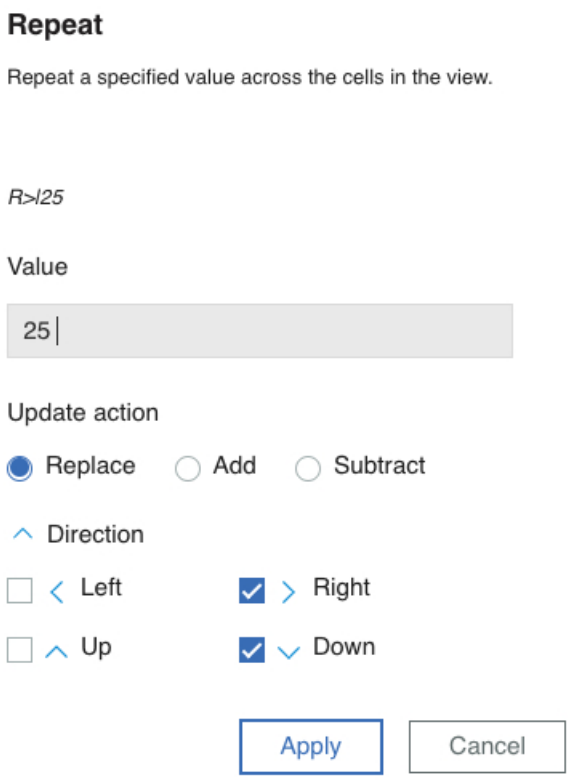
To spread the value 60 equally across cells to the right and down, type the following code:

```
S>|60
```

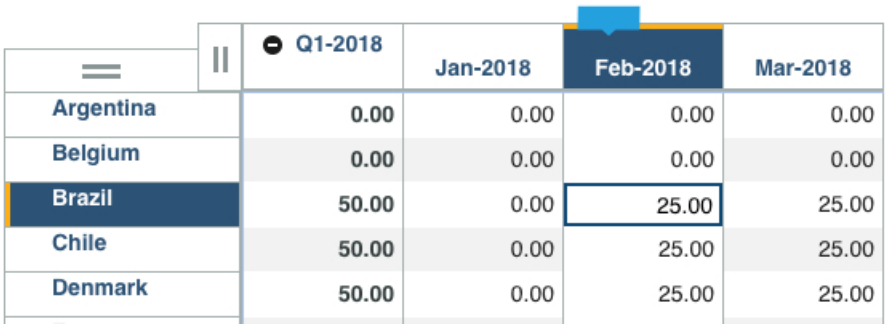
## Repeat data spreading method

The Repeat method repeats a specified value across cells in a view.

For example, right-click a cell at the intersection of Brazil and February, and select Repeat for the value 25 in the Right in the Right and Down directions:




This results in the following view:



| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 0.00 | 0.00 | 0.00 | 0.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 |
| Brazil | 50.00 | 0.00 | 25.00 | 25.00 |
| Chile | 50.00 | 0.00 | 25.00 | 25.00 |
| Denmark | 50.00 | 0.00 | 25.00 | 25.00 |

## Clear data spreading method

The Clear method clears values from cells in a view. You can apply this method to either leaf cells or consolidated cells. When you apply the Clear method to a consolidated cell, all leaves of the consolidation get set to zero.

If you initiate the Clear method from the selected cell and extend the spreading operation downward, the leaves of all consolidations below the point of insertion are cleared.

### Data entry shortcut code

To clear values to the left and down, type this code:

```
c<|
```

## Percent change data spreading method

The Percent Change method multiplies the current cell values by a specified percentage. The product of that multiplication can then replace, be added to, or be subtracted from the existing cell values.

For example, consider the following view that contains a range of values in increments of 10.

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 60.00 | 10.00 | 20.00 | 30.00 |
| Belgium | 150.00 | 40.00 | 50.00 | 60.00 |
| Brazil | 240.00 | 70.00 | 80.00 | 90.00 |

When you apply the Percent Change method to these cells and specify a % Change value of 10, the system multiplies each cell value by 10% (or .10). If you select the Add update action, the product of multiplication is added to the existing cell values. The result is that each cell value is incremented by 10%.

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 66.00 | 11.00 | 22.00 | 33.00 |
| Belgium | 165.00 | 44.00 | 55.00 | 66.00 |
| Brazil | 264.00 | 77.00 | 88.00 | 99.00 |

### Data entry shortcut code

To increase the value of each cell by 10% to the right and down, type this code:

```
P%+>|10
```

## Repeat leaves data spreading method

The Repeat Leaves method copies a specified value to the leaves (children) of a consolidation. When you apply this method, you can copy the value to all leaves of the consolidation or only to those leaves that already contain non-zero values.

For example, say there are several leaves of Year, Argentina with values.

If you use the Repeat Leaves method to copy the value 400 to the leaves of Year, Argentina currently populated with non-zero values, the value 400 is copied to all leaves that contained non-zero values.

If you initiate the Repeat Leaves method from a cell identified by more than one consolidated member, the specified value is copied to all leaves associated with the cell. For example, in the following view, the selected cell is identified by two consolidated members: Year and S Series Sedan.

| | Year | 1 Quarter | Jan | Feb | Mar |
|---|---|---|---|---|---|
| S Series Sedan | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| S Series 1.8 L Sedan | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| S Series 2.0 L Sedan | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| S Series 2.5 L Sedan | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| S Series 2.5 L Sedan 4WD | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| S Series 3.0 L Sedan | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

If you initiate Repeat Leaves spreading from the highlighted cell, the specified value is copied to all cells identified by the leaves of Year and the leaves of S Series Sedan. For instance, if you use Repeat Leaves to copy the value 25 to all leaves of the highlighted cell, the result is as follows.

| | Year | 1 Quarter | Jan | Feb | Mar |
|---|---|---|---|---|---|
| S Series Sedan | 2400.00 | 600.00 | 200.00 | 200.00 | 200.00 |
| S Series 1.8 L Sedan | 300.00 | 75.00 | 25.00 | 25.00 | 25.00 |
| S Series 2.0 L Sedan | 300.00 | 75.00 | 25.00 | 25.00 | 25.00 |
| S Series 2.5 L Sedan | 300.00 | 75.00 | 25.00 | 25.00 | 25.00 |
| S Series 2.5 L Sedan 4WD | 300.00 | 75.00 | 25.00 | 25.00 | 25.00 |
| S Series 3.0 L Sedan | 300.00 | 75.00 | 25.00 | 25.00 | 25.00 |
| S Series 3.0 L Sedan 4WD | 300.00 | 75.00 | 25.00 | 25.00 | 25.00 |
| S Series 3.4 L Sedan | 300.00 | 75.00 | 25.00 | 25.00 | 25.00 |
| S Series 3.4 L Sedan 4WD | 300.00 | 75.00 | 25.00 | 25.00 | 25.00 |

#### Data entry shortcut code

To copy the value of 25 to each leaf of a consolidation, whether populated or not, type the following code

```
LR*25
```

## Equal leaves data spreading method

The Equal leaves spreading method distributes a specified value equally across all leaves of a consolidated cell. When you apply this method, you can choose to distribute the value to all leaves of the consolidation or only to those leaves that already contain non-zero values.

In this example, there are several leaves of Year, Argentina with zero values.

If you use the equal leaves method to distribute the value 1200 to all the leaves of Year, Argentina, the result is as follows.

| | 2018 | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 | Q2-2018 | Apr-2018 | May-2018 |
|---|---|---|---|---|---|---|---|---|
| Argentina | 1200.00 | 300.00 | 100.00 | 100.00 | 100.00 | 300.00 | 100.00 | 100.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Brazil | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |
| Chile | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

If you initiate the Equal leaves spreading method from a cell identified by more than one consolidated member, the value is distributed to all leaves associated with the cell. For example, say the selected cell is identified by two consolidated members: Year and S Series Sedan.

If you initiate Equal leaves spreading from the highlighted cell, the specified value is distributed to all cells identified by the leaves of Year and the leaves of S Series Sedan. For instance, if you use Equal leaves spreading to distribute the value 1200 to all leaves of the highlighted cell, the result is as follows.

| | Year | 1 Quarter | Jan | Feb | Mar | 2 Quarter | Apr | May | Jun |
|---|---|---|---|---|---|---|---|---|---|
| S Series Sedan | 1200.00 | 300.00 | 100.00 | 100.00 | 100.00 | 300.00 | 100.00 | 100.00 | 100.00 |
| S Series 1.8 L Sedan | 150.00 | 37.50 | 12.50 | 12.50 | 12.50 | 37.50 | 12.50 | 12.50 | 12.50 |
| S Series 2.0 L Sedan | 150.00 | 37.50 | 12.50 | 12.50 | 12.50 | 37.50 | 12.50 | 12.50 | 12.50 |
| S Series 2.5 L Sedan | 150.00 | 37.50 | 12.50 | 12.50 | 12.50 | 37.50 | 12.50 | 12.50 | 12.50 |

When you initiate Equal leaves spreading from a cell identified by multiple consolidated members, the RAM requirements of the cube can increase significantly. Where more than 10,000 cells are affected by the Equal leaves method, a warning is displayed. In circumstances where more than one million cells are affected, the spreading operation does not execute.

#### Data entry shortcut code

To distribute the value of 1200 equally across all leaves of a consolidation, type this code:

```
LS*1200
```

## Straight line data spreading method

The Straight line data spreading method populates cube cells by linear interpolation between two specified endpoints.

For example: if you have a start value of 100 and an end value of 200, the midpoint value would be 150.

The following view shows the effect of Straight line spreading across a range of six cells with the start value of 100 and and an end value of 200.

| | Jan-2018 | Feb-2018 | Mar-2018 | Apr-2018 | May-2018 | Jun-2018 |
|---|---|---|---|---|---|---|
| Argentina | 100.00 | 120.00 | 140.00 | 160.00 | 180.00 | 200.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

With the start value of 100 and the end value of 200, the option populates the cells between the start and the end cells with values at equal intervals.

## Growth percent data spreading method

The Growth percent method takes an initial value as a starting point and sequentially increments all values in a selected range of cells by the specified percentage.

The following view shows the result of applying the Growth percent method to a range of six cells where the initial value is 100 and the growth percentage is 10%. This example uses the Replace data action.

| | Jan-2018 | Feb-2018 | Mar-2018 | Apr-2018 | May-2018 | Jun-2018 |
|---|---|---|---|---|---|---|
| Argentina | 100.00 | 110.00 | 121.00 | 133.10 | 146.41 | 161.05 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

The initial value of 100 displays in the cell identified by Jan-2018, Argentina. Applying the growth percentage of 10% to 100 gives 110, the value in Feb-2018, Argentina. Applying the growth percentage of 10% to 110 yields 121, the value in Mar-2018, Argentina.

## Relative proportional data spreading method

Use the Relative Proportional Spread method to spread values to the leaves of a consolidation cell proportional to the leaves of a reference cell.

The following example demonstrates relative proportional spreading where both the initial cell and the reference cell are in the same cube.

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 0.00 | 0.00 | 0.00 | 0.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 |
| Brazil | 100.00 | 10.00 | 20.00 | 70.00 |
| Chile | 0.00 | 0.00 | 0.00 | 0.00 |
| Denmark | 0.00 | 0.00 | 0.00 | 0.00 |

The reference cell can be located in the cube from which you initiate spreading or in a separate cube. However, the reference cell must share the same consolidations as the cell from which you initiate spreading.

The previous view shows a single consolidated value of 100 in the cell identified by 1 Quarter, Brazil. By looking at the leaves of 1 Quarter, you can see that:

- Jan contains the value 10, which is 10% of 100
- Feb contains the value 20, which is 20% of 100
- Mar contains the value 70, which is 70% of 100

If you initiate relative proportional spreading from 1 Quarter, Argentina and specify a Data Action of Replace when spreading the value 400, the leaves of 1 Quarter, Argentina are populated as proportional to the leaves of 1 Quarter, Brazil:

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 400.00 | 40.00 | 80.00 | 280.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 |
| Brazil | 100.00 | 10.00 | 20.00 | 70.00 |
| Chile | 0.00 | 0.00 | 0.00 | 0.00 |
| Denmark | 0.00 | 0.00 | 0.00 | 0.00 |

- Jan contains the value 40, which is 10% of 400
- Feb contains the value 80, which is 20% of 400
- Mar contains the value 280, which is 70% of 400

## Relative percent adjustment data spreading method

The Relative Percent Adjustment method spreads values to the leaves of a consolidation by applying a percentage adjustment to the leaves of a reference cell.

This method increments the values in the leaves of the reference cell by a specified percentage. The resulting values are then spread to the leaves of the consolidation from which you initiated spreading.

The reference cell can be located in the cube from which you initiate spreading or in a separate cube. However, the reference cell must share the same consolidations as the cell from which you initiate spreading.

The following example illustrates relative percent adjustment spreading where both the initial cell and the reference cell exist in the same cube.

Say you have a single consolidated value of 600 is in the cell identified by 1 Quarter, Brazil. The leaves of 1 Quarter would look like this:

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 0.00 | 0.00 | 0.00 | 0.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 |
| Brazil | 600.00 | 100.00 | 200.00 | 300.00 |
| Chile | 0.00 | 0.00 | 0.00 | 0.00 |
| Denmark | 0.00 | 0.00 | 0.00 | 0.00 |

- Jan contains the value 100
- Feb contains the value 200
- Mar contains the value 300

If you initiate Relative Percent Adjustment spreading from 1 Quarter, Argentina and specify a percent adjustment of 50% while using 1 Quarter, Brazil as a reference cell, the result is as follows.

| | Q1-2018 | Jan-2018 | Feb-2018 | Mar-2018 |
|---|---|---|---|---|
| Argentina | 300.00 | 50.00 | 100.00 | 150.00 |
| Belgium | 0.00 | 0.00 | 0.00 | 0.00 |
| Brazil | 600.00 | 100.00 | 200.00 | 300.00 |
| Chile | 0.00 | 0.00 | 0.00 | 0.00 |
| Denmark | 0.00 | 0.00 | 0.00 | 0.00 |

Note that the leaves of 1 Quarter, Argentina are populated with values that are 50% of the values found in the leaves of 1 Quarter, Brazil:

- Jan, Argentina contains the value 50, a 50% adjustment of the value in Jan, Brazil
- Feb, Argentina contains the value 100, a 50% adjustment of the value in Feb, Brazil
- Mar, Argentina contains the value 150, a 50% adjustment of the value in Mar, Brazil

## Exploration navigation shortcuts

You can use keystrokes to move to different locations in an exploration (table) view.

*Table 26. Exploration navigation shortcuts*

| Keystroke | Navigation result |
|---|---|
| **Up**, **down**, **left**, or **right arrow** keys | Moves to the adjacent cell in the direction represented by the arrow key. |
| **Backspace** | Clears the content of the active cell.<br>In cell editing mode, Backspace deletes the character to the left of the insertion point or deletes selected text. |

| Table 26. Exploration navigation shortcuts (continued) | |
|---|---|
| **Keystroke** | **Navigation result** |
| **Delete** | Clears the content of the active cell.<br>In cell editing mode, Delete deletes the character to the right of the insertion point or deletes selected text. |
| **Esc** | When in cell editing mode, Esc cancels the editing action. |
| **Ctrl + Home** | Moves to the first cell of the exploration. |
| **Ctrl + End** | Moves to the last cell of the exploration. |
| **Page Up** | Moves one screen up in the exploration. |
| **Alt + Page Up** | Moves one screen to the left in the exploration. |
| **Page Down** | Moves one screen down in the exploration. |
| **Alt + Page Down** | Moves one screen to the right in the exploration. |
| **F2** | Opens the current cell in editing mode. |
| **Ctrl + right arrow key** | Moves to the last visible cell in the current row. |
| **Ctrl + left arrow key** | Moves to the first visible cell in the current row. |
| **Ctrl + down arrow key** | Moves to the last visible cell in the current column. |
| **Ctrl + up arrow key** | Moves to the first visible cell in the current column. |

## Naming conventions

IBM Planning Analytics has the following naming conventions. Some of these conventions are enforced.

Although some of these characters are not reserved, it is a good practice to avoid the use of these special characters in most cases when you name objects and members. For more information, see "Member names and MDX expressions" on page 1266.

| Table 27. Special characters to avoid in object and member names | |
|---|---|
| **Character** | **Description** |
| ' | apostrophe |
| * | asterisk |
| @ | at sign - see "Object names in TM1 rules" on page 1266. |
| \ | back-slash |
| : | colon |
| , | comma |
| { | curly brace - see "The curly brace in object names" on page 1265. |
| " | double-quote |

| *Table 27. Special characters to avoid in object and member names (continued)* | |
|---|---|
| **Character** | **Description** |
| ! | exclamation mark - see “Object names in TM1 rules” on page 1266. |
| > | greater-than |
| < | less-than |
| - | minus sign - in member names. See “Member names and MDX expressions” on page 1266. |
| \| or ¦ | pipe or broken pipe |
| + | plus sign - in member names. See “Member names and MDX expressions” on page 1266. |
| ? | question-mark |
| ; | semicolon |
| / | slash |
| ~ | tilde - see “Object names in TM1 rules” on page 1266. |
| ^ | caret - see “Member names and MDX expressions” on page 1266. |

## Reserved characters per component

The following characters are explicitly reserved for the following components and must never be used when you name objects in these contexts:

- TM1 Architect reserves the following characters:

```
\ / : * ? " < > | }
```

- TM1 Server reserves these characters in these objects: Cube, Dimension, Subset, View, Process, Chores.

```
\ / : * ? " < > | ' ; ,
```

- For process variable names, the identifier cannot contain any special characters except for:

```
AllowableChars[] = ".$%_`";
```

## The curly brace in object names

It is a good practice to avoid the use of the right curly brace (}) as the first character in any user-created TM1 object name. TM1 control object names always begin with the right curly brace. If a user-created object name begins with a right curly brace, the object becomes hidden if the **Display Control Objects** parameter is turned off.

## Member names and MDX expressions

Do not use + or - as the first character of a member name. Although only the first member in a set when slicing to active form cannot use + or - as the first character in the element name, it is a good practice to never use + or - as the first character of an member name.

Do not use a ^ in a member name. The ^ character can be used as the delimiter between the ancestor and multi-parent member's name but when a member name that contains this character is referenced in an MDX expression, it cannot be escaped.

Although all the other characters available for use in member names are technically not restricted, it is good practice to avoid the special characters that are listed in the previous table when you name members.

A member name can contain a right square bracket ( ] ) but when a member name that contains this character is referenced in an MDX expression, the character needs to be escaped by doubling it. For example, a member that is named Array[N] Elements, can be referred to in an MDX expression as [Array[N]] Elements].

## Object names in TM1 rules

Although technically allowed, it is a good practice to avoid using these special characters in object names because they may conflict when used in a rules expression. This guideline protects you if the objects or members ever become part of a rule statement where those special characters are not permitted.

- For example, ) | ~ ; @ \ / : * ? " < > are all often found in rules statements and should not be used in object names.
- The @ is technically not restricted, however it is a good practice to avoid using the @ character in object names or member names because the @ character is also a string comparison operator in TM1 rules. If you reference any object with a name that contains the @ character in rules, the object name must be enclosed in single quotation marks. For example, a dimension named products@location must be referenced as 'products@location' in rules. Escaping the name with quotation marks does not work in all cases, so it is best to avoid the use of @ in all cases when naming objects.
- Escaping the special character using quotation marks does not work for ! or in certain rule expressions.
- The exclamation point ! character must not be used in object names because it is also used in rules expressions. For example:

```
DB('MarketExchange',!market,!date)
```

## Maximum string length for data directory and object names

The entire string that is represented by the combination of the TM1 Server data directory name and the object name is limited to 128 bytes. For example, if your data directory is C:\Financial data\TM1\ (22 bytes), object names are limited to 106 bytes, inclusive of a file extension such as .cub or .rux.

Some TM1 objects, such as views, subsets, and applications, are stored in subdirectories of the TM1 Server data directory. In this case, the 128-byte limit is applied to the combination of the TM1 Server data directory, the subdirectory, and the object name.

## Case sensitivity

Object names are not case-sensitive. For example, the dimension name actvsbud is equivalent to ActVsBud.

## Spaces in object names

Spaces are allowed in all object names, but spaces are ignored by the TM1 Server. The TM1 Server considers the dimension name Act Vs Bud to be equivalent to ActVsBud (or actvsbud).

### User names

User names that include reserved characters cannot save private objects.

# Map reference info

Planning Analytics Workspace supports specific geographic entity values and languages for maps.

## Supported languages for maps

Maps in Planning Analytics Workspace support continent, county, and local state and province names in a number of different languages.

Planning Analytics Workspace supports all continent and country names in the following list of languages. Planning Analytics Workspace also support local state and province names in their respective local languages for the languages on this list. For example, the name of the state of North Carolina in the United States is supported only in English. It is not supported in French as "Caroline du Nord" or in any other language.

Supported languages:

1. Catalan
2. Chinese-simplified (China)
3. Chinese-traditional (Taiwan)
4. Croatian
5. Czech
6. Danish
7. Dutch
8. English
9. Finnish
10. French
11. German
12. Greek
13. Hungarian
14. Italian
15. Japanese
16. Kazakh
17. Korean
18. Norwegian Bokmål
19. Polish
20. Portuguese (Brazil)
21. Romanian
22. Russian
23. Slovak
24. Slovenian
25. Spanish
26. Swedish
27. Thai
28. Turkish

# Troubleshooting Planning Analytics Workspace

You can use these topics to help you get the best experience in IBM Planning Analytics Workspace.

## Setting the TM1 Web session timeout

The default TM1 Web session timeout is 20 minutes. When TM1 websheets are deployed to IBM Planning Analytics Workspace, you might encounter TM1 Web session timeouts. You can modify this setting in your environment.

### About this task

When TM1 websheets are deployed to Planning Analytics Workspace, the recommended session timeout is 60 minutes.

**Note:** As of IBM Planning Analytics Local version 2.0.6, you must **not** change the `session-timeout` value in the `web.xml` file.

In IBM Planning Analytics Local version 2.0.6, there is a parameter in the `tm1web_config.xml` file called `HttpSessionTimeout`. You can use this parameter to customize the session timeout (in minutes) of the HTTP session for TM1 Web.

If the `HttpSessionTimeout` parameter is not specified (missing or blank), the value is less than 1 or not a numerical value, the default `session-timeout` that is defined in the `web.xml` file is used.

If you are using IBM Planning Analytics Local version 2.0.6 or later, to customize the session timeout for TM1 Web, set the `HttpSessionTimeout` parameter in `tm1web_config.xml`. See step "1" on page 1268.

If you are using IBM Planning Analytics Local version 2.0.5 or earlier, to change the default session timeout, set the `<session-timeout>` parameter in `web.xml`. See step "2" on page 1268.

### Procedure

1. To customize the session timeout, follow these steps.
   a) Open the`tm1web_config.xml` in a text editor.
      The `tm1web_config.xml` file is located in your `<pa_installation_directory>\webapps\tm1web\WEB-INF\configuration` directory.
      For example, `C:\Program Files\IBM\cognos\tm1_64\webapps\tm1web\WEB-INF\configuration`.
   b) Change the `HttpSessionTimeout` to 60 or a value that is required by your environment.

```
<add key="HttpSessionTimeout" value="60" />
```

   c) Save and close the `tm1web_config.xml`.
2. If you are using IBM Planning Analytics Local version 2.0.5 or earlier, to change the default session timeout, follow these steps.
   a) Open `web.xml` in a text editor.
      The `web.xml` file is located in your `<pa_installation_directory>\webapps\tm1web\WEB-INF` directory.
      For example, `C:\Program Files\IBM\cognos\tm1_64\webapps\tm1web\WEB-INF`.
   b) Change the `<session-timeout>` value to 60 or a value that is required by your environment.

```
<session-config>
    <session-timeout>20</session-timeout>
</session-config>
```

   c) Save and close `web.xml`.
3. Restart the IBM TM1 Application Server service.

## Slow scrolling in Google Chrome browser

Scrolling in the Planning Analytics Workspace data tree or in cube views can be slow or unresponsive when using the default Chrome browser setting for Smooth Scrolling. To resolve this issue, modify the Smooth Scrolling setting from **Default** to **Disabled**.

### Procedure

1. Type `chrome://flags` in the Chrome address bar.
2. Scroll down to find the **Smooth Scrolling** setting.
3. Change the setting from **Default** to **Disabled**.
4. Restart Chrome.

## Help error: Unable to get resource file

When you view a topic in the IBM Planning Analytics Workspace integrated help system, you might see an error message:`Unable to get resource file - https://www.ibm.com/support/knowledgecenter/SSD29G_2.0.0/com.ibm.swg.ba.cognos.tm1_prism_gs.2.0.0.doc/`*`filename`*.

### About this task

If this message appears, there may be a problem with the Knowledge Center.

### Procedure

Try one of the following solutions:

- Wait a few minutes and then try selecting the topic again.
- Copy and paste the URL in the error message into your browser.

# Accessibility

Accessibility features help users who have a physical disability, such as restricted mobility or limited vision, to use information technology products successfully.

The major accessibility features are described in the following list:

- You can use the keyboard to work in the conditional formatting editor. To find out more, see "Use the keyboard to work with conditional formatting" on page 627.
- You can use command keys, or shortcut keys, to navigate through IBM Planning Analytics Workspace. Shortcut keys directly trigger an action and usually use the CTRL or Cmd keys. For example, to save a workbook, use CTRL+S.
- Planning Analytics Workspace uses Web Accessibility Initiative - Accessible Rich Internet Applications (WAI-ARIA). This means that people with limited vision can use screen-reader software, along with a digital speech synthesizer, to listen to what is displayed on the screen.
- You can use a high-contrast display in Planning Analytics Workspace.

For more information about the commitment that IBM has to accessibility, see IBM Accessibility on the web (www.ibm.com/able).

# Notices

This information was developed for products and services offered worldwide.

This material may be available from IBM in other languages. However, you may be required to own a copy of the product or product version in that language in order to access it.

IBM may not offer the products, services, or features discussed in this document in other countries. Consult your local IBM representative for information on the products and services currently available in your area. Any reference to an IBM product, program, or service is not intended to state or imply that only that IBM product, program, or service may be used. Any functionally equivalent product, program, or service that does not infringe any IBM intellectual property right may be used instead. However, it is the user's responsibility to evaluate and verify the operation of any non-IBM product, program, or service. This document may describe products, services, or features that are not included in the Program or license entitlement that you have purchased.

IBM may have patents or pending patent applications covering subject matter described in this document. The furnishing of this document does not grant you any license to these patents. You can send license inquiries, in writing, to:

IBM Director of Licensing
IBM Corporation
North Castle Drive
Armonk, NY 10504-1785
U.S.A.

For license inquiries regarding double-byte (DBCS) information, contact the IBM Intellectual Property Department in your country or send inquiries, in writing, to:

Intellectual Property Licensing
Legal and Intellectual Property Law
IBM Japan Ltd.
19-21, Nihonbashi-Hakozakicho, Chuo-ku
Tokyo 103-8510, Japan

The following paragraph does not apply to the United Kingdom or any other country where such provisions are inconsistent with local law: INTERNATIONAL BUSINESS MACHINES CORPORATION PROVIDES THIS PUBLICATION "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. Some states do not allow disclaimer of express or implied warranties in certain transactions, therefore, this statement may not apply to you.

This information could include technical inaccuracies or typographical errors. Changes are periodically made to the information herein; these changes will be incorporated in new editions of the publication. IBM may make improvements and/or changes in the product(s) and/or the program(s) described in this publication at any time without notice.

Any references in this information to non-IBM Web sites are provided for convenience only and do not in any manner serve as an endorsement of those Web sites. The materials at those Web sites are not part of the materials for this IBM product and use of those Web sites is at your own risk.

IBM may use or distribute any of the information you supply in any way it believes appropriate without incurring any obligation to you.

Licensees of this program who wish to have information about it for the purpose of enabling: (i) the exchange of information between independently created programs and other programs (including this one) and (ii) the mutual use of the information which has been exchanged, should contact:

IBM Software Group
Attention: Licensing

© Copyright IBM Corp. 2009, 2025

3755 Riverside Dr.
Ottawa, ON
K1V 1B7
Canada

Such information may be available, subject to appropriate terms and conditions, including in some cases, payment of a fee.

The licensed program described in this document and all licensed material available for it are provided by IBM under terms of the IBM Customer Agreement, IBM International Program License Agreement or any equivalent agreement between us.

Any performance data contained herein was determined in a controlled environment. Therefore, the results obtained in other operating environments may vary significantly. Some measurements may have been made on development-level systems and there is no guarantee that these measurements will be the same on generally available systems. Furthermore, some measurements may have been estimated through extrapolation. Actual results may vary. Users of this document should verify the applicable data for their specific environment.

Information concerning non-IBM products was obtained from the suppliers of those products, their published announcements or other publicly available sources. IBM has not tested those products and cannot confirm the accuracy of performance, compatibility or any other claims related to non-IBM products. Questions on the capabilities of non-IBM products should be addressed to the suppliers of those products.

All statements regarding IBM's future direction or intent are subject to change or withdrawal without notice, and represent goals and objectives only.

This information is for planning purposes only. The information here is subject to change before the products described become available.

This information contains examples of data and reports used in daily business operations. To illustrate them as completely as possible, the examples include the names of individuals, companies, brands, and products. All of these names are fictitious and any similarity to the names and addresses used by an actual business enterprise is entirely coincidental.

COPYRIGHT LICENSE:

This information contains sample application programs in source language, which illustrate programming techniques on various operating platforms. You may copy, modify, and distribute these sample programs in any form without payment to IBM, for the purposes of developing, using, marketing or distributing application programs conforming to the application programming interface for the operating platform for which the sample programs are written. These examples have not been thoroughly tested under all conditions. IBM, therefore, cannot guarantee or imply reliability, serviceability, or function of these programs. The sample programs are provided "AS IS", without warranty of any kind. IBM shall not be liable for any damages arising out of your use of the sample programs.

Each copy or any portion of these sample programs or any derivative work, must include a copyright notice as follows:

© (your company name) (year). Portions of this code are derived from IBM Corp. Sample Programs. © Copyright IBM Corp. _enter the year or years_.

If you are viewing this information softcopy, the photographs and color illustrations may not appear.

This Software Offering does not use cookies or other technologies to collect personally identifiable information.

## Product Information

This document applies to IBM Planning Analytics version 2.0.0 and may also apply to subsequent releases.

## Copyright

Licensed Materials - Property of IBM

© Copyright IBM Corp. 2007, 2022.

US Government Users Restricted Rights – Use, duplication or disclosure restricted by GSA ADP Schedule Contract with IBM Corp.

IBM, the IBM logo, and ibm.com are trademarks or registered trademarks of International Business Machines Corp., registered in many jurisdictions worldwide. Other product and service names might be trademarks of IBM or other companies. A current list of IBM trademarks is available on the web in " Copyright and trademark information " at www.ibm.com/legal/copytrade.shtml.

## Other trademarks

The following terms are trademarks or registered trademarks of other companies:

- Microsoft, Windows, Windows NT, and the Windows logo are trademarks of Microsoft Corporation in the United States, other countries, or both.
- Adobe, the Adobe logo, PostScript, and the PostScript logo are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States, and/or other countries.
- The registered trademark Linux is used pursuant to a sublicense from the Linux Foundation, the exclusive licensee of Linus Torvalds, owner of the mark on a worldwide basis.
- UNIX is a registered trademark of The Open Group in the United States and other countries.
- Java and all Java-based trademarks and logos are trademarks or registered trademarks of Oracle and/or its affiliates.
- Red Hat, JBoss®, OpenShift, Fedora®, Hibernate®, Ansible®, CloudForms®, RHCA®, RHCE®, RHCSA®, Ceph®, and Gluster® are trademarks or registered trademarks of Red Hat, Inc. or its subsidiaries in the United States and other countries.

Microsoft product screen shot(s) used with permission from Microsoft.

