Oracle Financial Services Model Management and Governance

User Guide

Release 8.1.2.3.0

November 2022

ORACLE
Financial Services

ORACLE

OFS Model Management and Governance User Guide

Document Control

Version Number	Revision Date	Change Log
1.0	Feb 2022	Created the user guide for release 8.1.2.0.0.
2.0	June 2022	Created the user guide for release 8.1.2.1.0.
3.0	August 2022	Created the user guide for release 8.1.2.2.0.
4.0	November 2022	Created the user guide for release 8.1.2.3.0.

Table of Contents

1 Preface

This preface provides information for the Oracle Financial Services Model Management and Governance application (OFS MMG) User Guide.

1.1 Access to Oracle Support

Oracle customers have access to electronic support through My Oracle Support. For information, visit:

- http://www.oracle.com/pls/topic/lookup?ctx=acc&id=info
- http://www.oracle.com/pls/topic/lookup?ctx=acc&id=trs if you are hearing impaired.

1.2 Audience

This document is intended for the system administrators and users configuring the workspace and models in OFS MMG.

1.3 What is New in This Release

1. **Workspace**
   - Download option is introduced.
   - Workspace Delete now additionally purges underlying objects.
2. **API Configuration**
   - Support for registering third-party APIs**.**
   - Ability to map and integrate with Model Pipelines.
3. **UI/UX Enhancements**
   - Improved UI messages and logging.
   - Standardized layouts and controls on all the screens.
   - UI/UX improvements in line with the Redwood design.
4. **Health UI**
   - Introduced status messages related to Graphs and PGX server.
   - Three new tabs are introduced:
     - Service Status: Provides information on the services that are running.
     - Environment: Provides information on the System and Application properties.

- Log Trace: Provides the logs which will help during analysis and debugging.

5. **Model Pipelines**
   - Improvements on Session Management.
   - Added new tasks to support API configurations, Model Builder, Graphs, and Kafka (Beta) in the Pipeline Canvas screen.
   - Ability to Clone Parameter Sets.
   - Ability to attach and promote Parameter Sets against a Model.
   - The Pipeline execution statuses are now displayed on the header in the Pipeline Canvas screen.
   - Ability to execute the Models in another session with different parameters simultaneously.
   - A new icon **Open in notebook session** has been added which displays the current status of the execution.
   - Canvas View has been introduced in the Execution History screen.
   - The execution statuses are now displayed in the Execution History screen.
   - Compare option is introduced which allows you to compare the executions.
   - Run Stats has been added in the Simulations screen which provides information on the Start and End time of the execution.
6. **Scheduler Service**
   - Ability to schedule MMG batches through external scheduler (ESIC).
   - Batch status report generation.
   - Improved model execution logs for tracking and debugging.
7. **Model Training**
   - Ability to build a Catalog for a Model.
   - Technique registration and Model building.
   - Integration with Model Pipelines. You can now promote the associated trained Models to production.
   - Templates and out-of-the-box seeded Libraries and Techniques are made available.
   - Support for Open Neural Network Exchange Library (ONNX) for Modeling.
8. **Graphs**

   Unlike traditional Relational Database Management Systems, the Graph Pipeline feature allows you to view the data relationships in a graphical format. This feature uses the latest technology to harness the power of Graph Analytics to give Financial Institutions the ability to monitor the Data Financial Institutions, effectively.

   For more information, see the Graph Pipeline section.

- Graph Refresh Schedules feature is included inside the Graphs.
- UI/UX improvements on the Graph screens.
- Integration with the Model Pipelines. You can now promote the associated Graphs and its execution schedules to production.
- PGX and related interpreters have been added to the Data Studio.

9. **Performance Improvements**
   - Responsiveness of Data Studio.
   - Loading for Dataset's Entity selection.
   - Configurable performance parameters for Data Studio.
10. **Dataset**
    - Monitor the data variations over time.
    - Analyze and compare the Data snapshots.
    - Generate Data Drift report.
11. **Third-Party Library upgrades**
    - Java has been upgraded to version 11 and above. Any version less than 11 is not supported.
    - OJET has been upgraded to version 13.
    - Data Studio has been upgraded to 22.4.2.
    - PGX server version 22.4.2 has been added to support the Graph feature.
12. **Enhancements to Installer and Utility**
    - Additional support to approve the models from Model Import Utility.
    - Enhanced the Installer where you can enable or disable the R interpreter in the Data Studio.

1.4 Additional Resources

This section identifies additional resources for the OFS MMG Application Pack. You can access the online documentation for the OFS MMG 8.1.2.3.0 from the Oracle Help Center (OHC).

- OFS Model Management and Governance Release Notes
- OFS Model Management and Governance Installation Guide

To find additional information about how Oracle Financial Services solves real business problems, see our website at www.oracle.com/financialservices.

1.5 Conventions Used

The following table lists the conventions used in this guide.

Table 1: Conventions Used in this Guide

Convention	Meaning
Italics	• Names of books, chapters, and sections as references • Emphasis
Bold	• The object of an action (menu names, field names, options, button names) in a step-by-step procedure • Commands typed at a prompt • User input
`Monospace`	• Directories and subdirectories • File names and extensions • Process names • Code sample, including keywords and variables within the text and as separate paragraphs, and user-defined program elements within the text
Hyperlink	Hyperlink type indicates the links to external websites, internal document links to sections.

1.6 Acronyms Used

The following table lists the acronyms used in this guide.

Table 2: Acronyms Used in this Guide

Conventions	Description
BA	Business Analysts
Infodom	Information Domain
Navigation Tree Menu or Navigation Menu	Left-hand side menu
OFS AAI	Oracle Financial Services Analytical Application Infrastructure
OFSAA	Oracle Financial Services Analytical Applications
OFS MMG	Oracle Financial Services Model Management and Governance
Production Infodom	Production Information Domain
Sandbox Infodom	Sandbox Information Domain
SA	System Administrator
URL	Uniform Resource Locator
UI	User Interface
PGX	Parallel Graph AnalytiX

2 Getting Started

This section gives an introduction to OFS Model Management and Governance and the requirements to use the application.

2.1 About this Guide

This guide has information about configuring and publishing workspaces using the OFS Model Management and Governance application. The further sections in this guide provide information for the creation and deployment of models into production. Use it for information about workspace and model management or help with the processes in the application.

2.2 About Oracle Financial Services Model Management and Governance

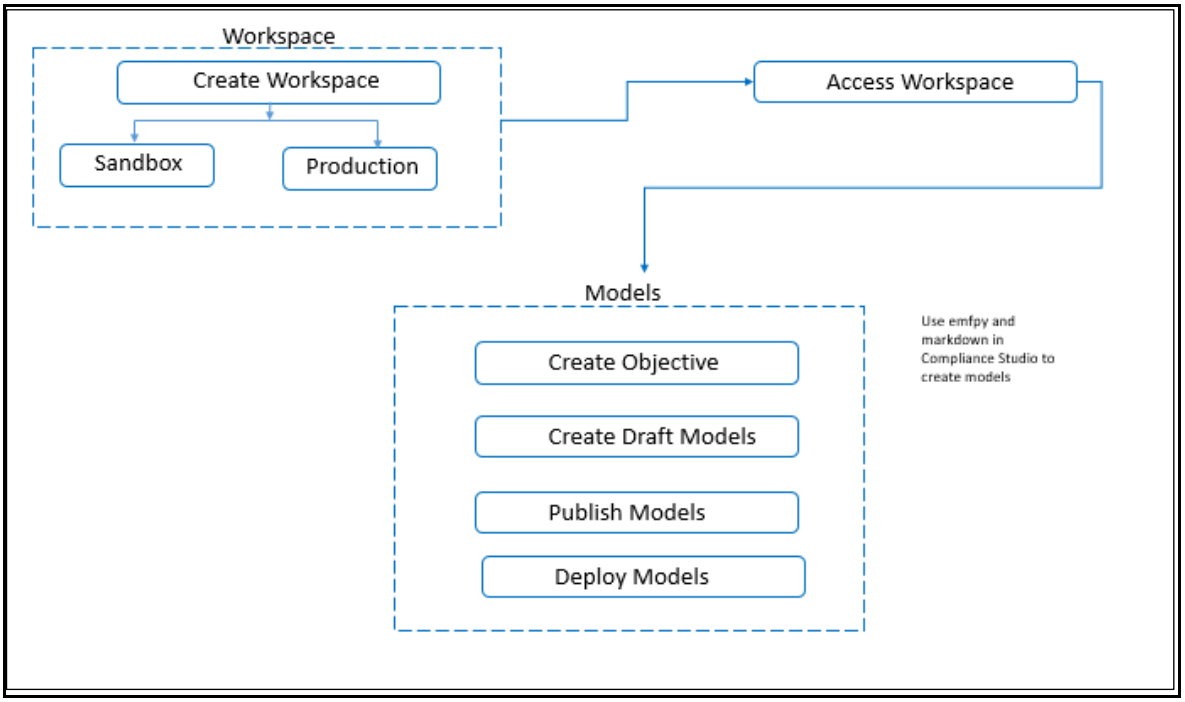
Financial institutions require models that work on traditional statistical techniques, modern machine-learning methods, computational and simulation models. Oracle Financial Services Model Management and Governance leverage the Data Studio environment to develop, deploy, and manage models at the enterprise level.

The OFS Model Management and Governance application enables institutions to implement their IT policies while providing flexibility and freedom that Data Scientists and Statistical Modelers desire. OFS MMG's design facilitates financial institutions to manage external regulatory and internal governance policies by building testing models in a workspace environment. A workspace is provisioned and authorized for use (usually by an administrator) before making it available to modelers. Administrative users grant analysts and modelers access to workspaces along with a subset of production data to build models. Validated and approved models can then be promoted from workspaces to the enterprise model repository. Models in the repository can then be woven into analytical application flows crafted by mixing data management tasks, model execution, and deterministic business logic.

2.3 Oracle Financial Services Model Management and Governance Workflow

The workflow involves the creation of workspaces and the creation of Models mapped to the Workspaces. Models are then configured as training models that you can use to perform model visualizations and test for the outcomes. You can then publish a model into production and make it available to users after you have determined that the models and the parameters used to construct the models meet the requirements of your business logic.

Figure 1: Oracle Financial Services Model Management and Governance Workflow



2.4 Components of Oracle Financial Services Model Management and Governance

The following are the components of Oracle Financial Services Model Management and Governance application:

- Workspace Management
- Model Management
- Dataset
- Model Pipelines
- Model Actions
- Graphs
- Scheduler Service
- Audit Trail
- Data Studio Options
- Object Migration
- Model Training

2.4.1 Workspaces

Workspace Management is where the workspace Administrators define datasets and make them available to modelers in Oracle Financial Services Model Management and Governance. It is an environment where the data is prepared for modelers to use in their modeling creation activity.

Workspaces are provisioned with the data required for modeling by administrators, who configure workspaces with subsets of production data. The data in the workspace is made available to modelers using datasets, who then build models with the dataset without any exposure being provided to the physical data tables and columns in the database. In effect, the data is ready for the modelers, and they do not have to undergo the arduous task of accessing and querying the database. See the Using Workspaces section for information on how to use Workspace Management in the application.

2.4.2 Dataset

Datasets allow for the creation of a dataframe, a data structure that captures the logic that organizes data into a 2-dimensional table of rows and columns. Datasets allow for the reuse of these dataframes across models, as well as the features derived from the data source(s).

The Dataset allows you to manage the entire operation related to data set. You can perform the following two things using the Dataset window:

- Define a metadata on how you want to create a dataset
- In addition, a mechanism where you can take a snapshot of real time data and store it. So, it can be used later in the pipeline.

For more information, see the Dataset section.

2.4.3 Model Pipelines

Modeling refers to the process of designing a prototype based on a structured data model, for statistical analysis and to simulate real events and processes. Models in a workspace can be created or modified by a user with access to the workspace. Model versions are preserved in the workspace along with execution and output histories. Once a model has been validated in the workspace and considered fit for use, modelers can request to push the model into the production environment.

See the Using Model Pipeline section to create models from pre-defined models to predict business trends and validate the existing models.

2.4.4 Model Actions

The Model Actions window allows you to view the list of models for which the Workflow action has been taken which requires review or approve across the workspace.

For more information, see the Model Actions section.

2.4.5 Graphs

Unlike traditional relational database management systems, the Graph Pipeline feature allows you to view the data relationships in a graphical format.

This feature uses the latest technology to harness the power of Graph Analytics to give Financial Institutions the ability to monitor the data financial institutions effectively. The data is organized as nodes, relationships, and properties (property data is stored on the nodes or relationships). The results of analytics algorithms are stored as transient properties of nodes and edges in the Graph.

For more information, see the Graph Pipeline section.

2.4.6 Scheduler Service

The Scheduler Service is a service in the Infrastructure system that automates behind-the-scenes work that is necessary to sustain various enterprise applications and functionalities. This automation helps the applications to control unattended background jobs program execution.

The Scheduler Service contains a graphical user interface and a single point of control for the definition and monitoring of background executions.

For more information, see the Scheduler Service section

2.4.7 Audit Trail

The Audit Trail window provides the complete details of model. This shows the information such as, when Model was created, who created the Model, workflow of Model, for example when this Model became champion or deployed, and so on.

For more information, see the Audit Trail section.

2.4.8 Data Studio Options

MMG contains an underlying Notebook Server which has the following configurable options:

- Interpreters
- Tasks
- Permissions
- Credentials
- Templates

For more information, see the Data Studio Options section.

2.4.9 Object Migration

Object Migration is the process of migrating or moving objects between environments.

You may want to migrate objects for several reasons such as managing global deployments on multiple environments or creating multiple environments so that you can separate the development, testing, and production processes.

For more information, see the Object Migration section.

2.4.10 Model Training

The Model Training page allows you to add and manage the Model Technique, Model Library, and Model Objectives.

For more information, see the Model Training section.

2.5 Mapping User Groups

Users must be mapped to User Groups that are mapped to the required roles to access OFS MMG. The following subsections provide information about the user groups and roles required in addition to the information about configuring the user groups.

2.5.1 User Groups

The following table gives details about the User Groups in the Oracle Financial Services Model Management and Governance application.

Table 3: User Groups

User Group Name	User Group Description
IDNTYADMN	Identity Administrator group
IDNTYAUTH	Identity Authorizer group
MDLREV	The Modeling Reviewer Group. Users mapped to this group have access to the menu items in the OFS COMPLIANCE STUDIO application that are related to model review activities.
MDLAPPR	The Modeling Approver Group. Users mapped to this group have the rights to approve models created by the users.
MDLBATCHUSR	The Modeling Batch User. Scheduler can use this Group for executing batches.
WKSPADMIN	The Workspace Administrator Group. Users mapped to this group have access to all the menu items in the OFS COMPLIANCE STUDIO application. Additionally, they have authorization rights to create and populate workspaces.
MDLUSR	The Modeling User Group. Users mapped to this group have access to all the menu items in the OFS COMPLIANCE STUDIO application that is related to model creation.

User Group Name	User Group Description
DSUSRGRP	General Role User don't have access to modify Interpreter configurations. Users mapped to this group have access to all the menu items in the OFS COMPLIANCE STUDIO application that is related to model creation.
DSREDACTGRP	Roles for applying redaction in graph. This group will be applicable to only those users for whom graph redaction is required.

NOTE

At the first-time login, User Group mappings are initialized from AAI/IDCS for the newly provisioned users. These will be reflected in MMG Admin Console in next MMG login.

If User Group mappings are deleted in AAI/IDCS, it would not delete in MMG Admin Console. Admin needs to delete this in MMG Identity screens too.

Only the group with MDLSUMM role will be displayed in the Workspace provisioning steps.

MDLSUMM function is mapped to the MDLACCESS role.

2.5.2 User Group - Role Mapping

Map the user groups in the application to the roles in the following table to enable access to the OFS MMG application.

Table 4: User Group to Role Mapping

Group Name	Role Name
DSREDACTGRP	DSREDACT
IDNTYADMN	Batch Advance Role
IDNTYADMN	Batch Write Role
IDNTYADMN	Admin Link Role
IDNTYADMN	User Advanced Role
IDNTYADMN	Group Advanced Role
IDNTYADMN	Role Advanced Role
IDNTYADMN	Function Advanced Role
IDNTYAUTH	Group Authorize Role
IDNTYAUTH	User Authorize Role
IDNTYAUTH	Group Read Role
IDNTYAUTH	Admin Link Role

Group Name	Role Name
IDNTYAUTH	Function Read Role
IDNTYAUTH	Role Read Role
IDNTYAUTH	Role Authorize Role
MDLAPPR	DSINTER
MDLAPPR	Model Authorize
MDLAPPR	Model Deployment
MDLAPPR	Workspace Read
MDLAPPR	Model Read
MDLAPPR	Model Access
MDLAPPR	Workspace Access
MDLAPPR	DSAPPROVER
MDLBATCHUSR	DSBATCH
MDLREV	Workspace Read
MDLREV	Model Review
MDLREV	Model Access
MDLREV	Workspace Access
MDLREV	DSUSER
MDLREV	Model Read
MDLUSR	Model Advanced
MDLUSR	Model Write
MDLUSR	Model Read
MDLUSR	Batch Advance Role
MDLUSR	Model Execute
MDLUSR	DSUSER
MDLUSR	Model Access
MDLUSR	Workspace Access
MDLUSR	Workspace Read
MDLUSR	Datasource Access
MDLUSR	Datasource Write
MDLUSR	Datasource Read
WKSPADMIN	Workspace Access
WKSPADMIN	DSADMIN
WKSPADMIN	Identity MGMT advanced

Group Name	Role Name
WKSPADMIN	Workspace Authorize
WKSPADMIN	Workspace Read
WKSPADMIN	Workspace Write
DSUSRGRP	DSADMIN
GRAPHUSER	Graph Administrator
GRAPHUSER	Graph Read Role
GRAPHUSER	Graph Read Role
GRAPHUSER	Graph Execute Role
GRAPHADMINISTRATOR	Graph Administrator Role

2.6 Signing into the Oracle Financial Services Model Management and Governance Application

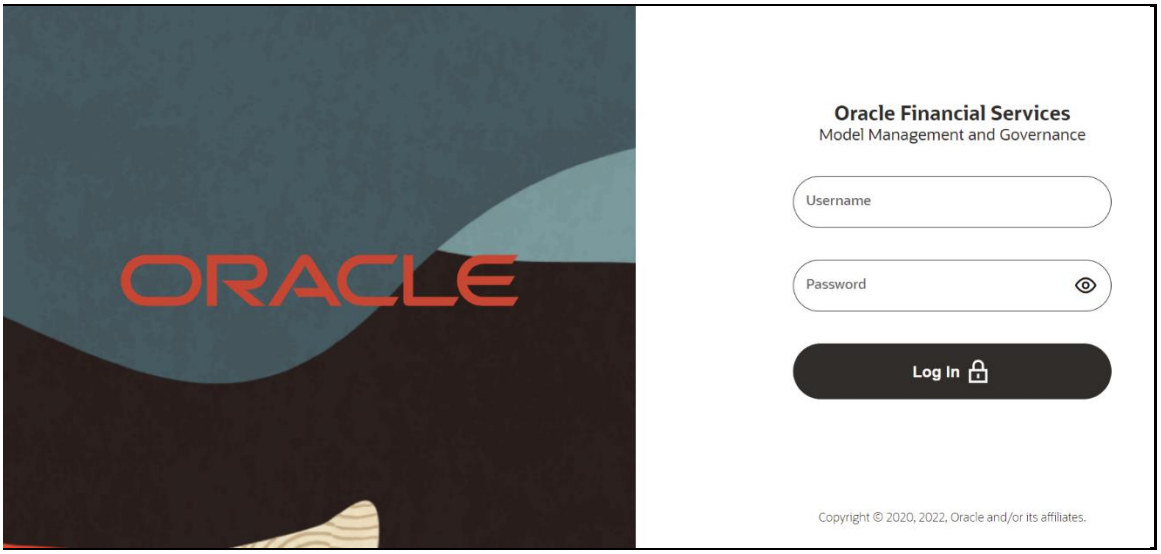
After the application is installed and configured, you can access the Oracle Financial Services Model Management and Governance application.

To access Oracle Financial Services Model Management and Governance, follow these steps:

1. Enter the application URL in your browser.

   The **Login** page is displayed.

   Figure 2: OFS Model Management and Governance Application Login Page



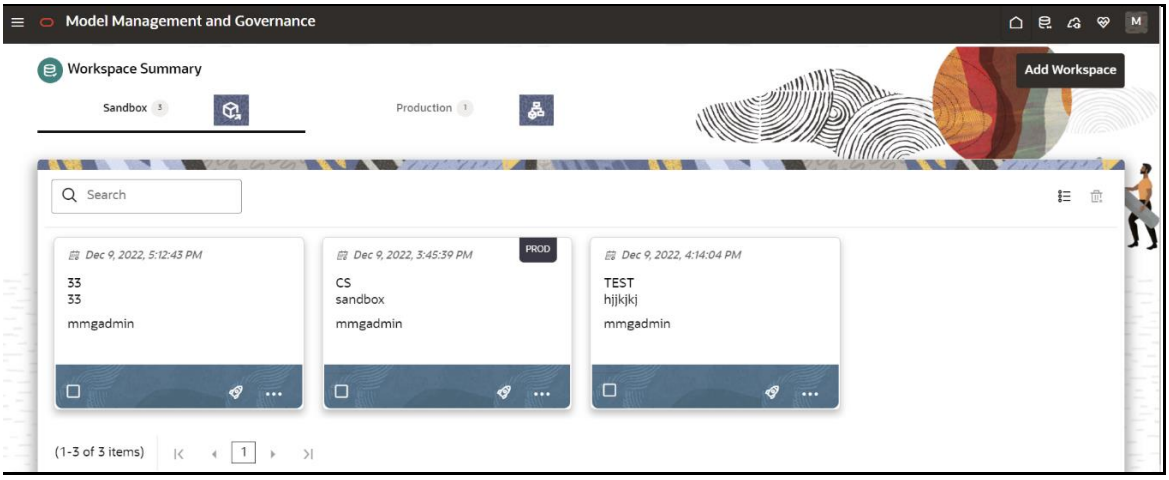
2. Enter your **Username** and **Password**.

3. Click **Log in**.

   The **Workspace Summary** page is displayed.

> **NOTE** If you are logged in for the first time or a new user, you must refresh the MMG URL which will enable the role validation.

Figure 3: Workspace Summary



2.7 Using the Application UI Features

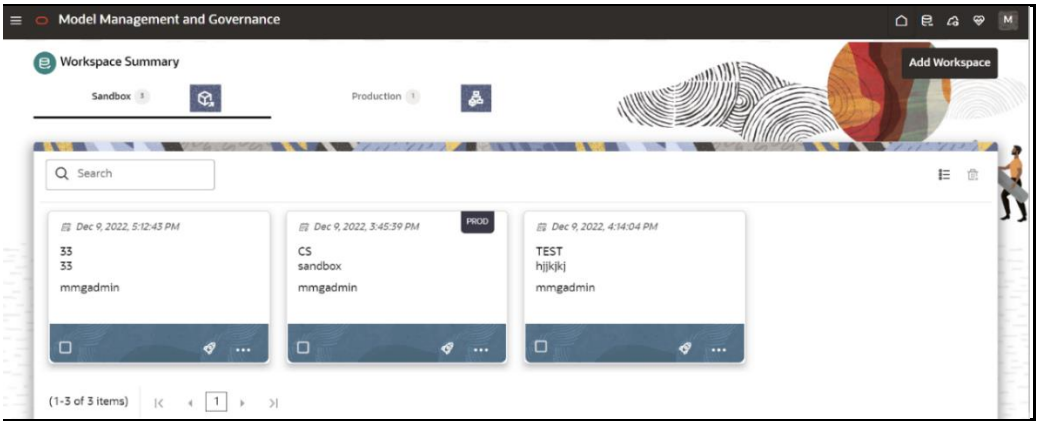
This section describes the user-interface of the Oracle Financial Services Model Management application. These are the common features that are found across the modules of Oracle Financial Services Model Management. It describes the organization of the user interface and provides step-by-step instructions for navigating through the application.

2.7.1 Home Page Components

The Home Page contains the following sections:

- Navigation Menu
- Home
- Data Source Summary
- API Configurations
- Health
- Profile
- Workspace Summary

Figure 4: Workspace Summary



2.7.1.1 Navigation Menu

Launch a Workspace, click **Navigation Menu** icon to navigate to Dashboard, Dataset, Advance Options, and other sections of MMG application.

2.7.1.2 Home

Click **Home** icon to navigate to **Workspace Summary** page from any other window in MMG application**.** You can add and manage Workspaces from this page

2.7.1.3 Data Source Summary

Click **Data Source Summary** icon to navigate to **Data Source Summary** page from any other window in MMG application**.** You can add and manage Data Sources from this page.

2.7.1.4 API Configurations

Click **API Configurations** icon to navigate to **API Summary** page from any other window in MMG application**.** You can configure and manage APIs from this page.

2.7.1.5 Health

Click **Health** icon to view the application status and version details**.** The Health page consists of three tabs:

- **Service Status**: Provides the status information on the services that are running such as Data Studio, Data Pipeline, and Graph Services.
- **Environment**: Provides information on the System and Application properties.
- **Log Trace**: Provides the logs which will help during analysis and debugging.

Clicking on the **Refresh**  icon will refresh the application status details.

2.7.1.6 Profile

The Header displays icons, buttons, and text for generic information and access to the OFSAA application's features. The following user-interface elements are displayed for OFS MMG:

- **User Name:** Displays the logged-in user name. Click this to select from the following options in the drop-down list:
  - **Admin:** Navigates to IDCS page.
  - **Logout:** Select to log out of the application.

2.7.1.7 Workspace Summary

Shows the list of Workspaces. Select the checkbox next to the Workspace that you want to View, Edit, Copy, or Delete. Click **Add Workspace** to create a new Workspace. You can search a Workspace using Workspace Code, Owner, Creation Date, or Workspace Type fields. The search list shows the Workspace details with Workspace Code, Owner, Creation Date, and Type.

2.8 Using the OFS MMG Application

The OFS MMG application displays windows and pages that are interconnected to the OFS AAI. Primarily, you will create Workspace. Within these Workspaces, you will create models, compare them for the best fit, and promote one to production.

Use the information in the following topics to create the workspaces and models:

- Using Workspaces
- Using Model Pipelines

The prerequisites for OFS MMG are as follows:

- To create a workspace, your user profile must be mapped to the Workspace Administrator role. For more information, see the User Groups section.
- To use a workspace to create models, your user profile must be mapped to the Model User role (such as a modeler). For more information, see the User Group - Role Mapping section.

2.9 Access the Workspace Dashboard Window

The Workspace Dashboard window allows you to view the models of launched workspace.

To access the Dashboard window, follow these steps:

1. Navigate to **Workspace Summary** page.

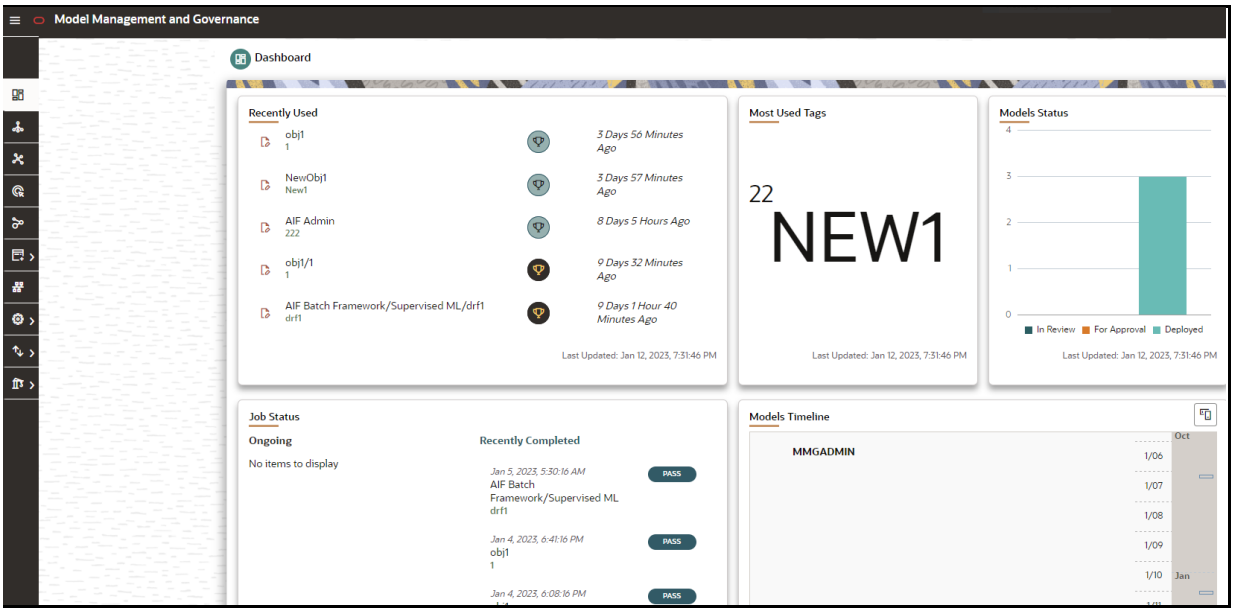
   The page displays workspace records in a table.

2. Click  next to corresponding Workspace to Launch Workspace.

   The MMG **Dashboard** window is displayed with application configuration and model creation menu.

The Workspace Dashboard shows the following details of a launched workspace:

- Recently Used
- Most Used Tags
- Models Status
- Job Status
- Models Timeline



2.10 Executing the Production Workspace Models from Analytical Applications Infrastructure

Prerequisites

- To execute the MMG Production Workspace Models from AAI, log in to My Oracle Support, and search for **33603448** patch under the **Patches & Updates** tab, download and apply it on the OFSAAI sever.
- Ensure the AAI Infodom and MMG Workspace names are same before you execute the Production Workspace Models.

To execute the MMG Production Workspace Models from the AAI application, perform the following steps:

1. In the AAI configuration schema, update the value for `MMG_SERVICE_URL` in the `nextgenemf_config` table with the values in the MMG installation.

   Example:

   `MMG_SERVICE_URL :<FQDN/IP>:<mmg_be_port>/<mmg_context_name>`

   > **NOTE** If the target MMG is using secured connection, import the target server certificate to the AAI application java keystore and restart the services.

2. Restart the AAI services.

   A new component STUDIOMODEL will be available in ICC, RRF, and PMF modules for selecting the Production Workspace Models of MMG.

   For more details, refer to the *Component: STUDIOMODEL section of* AAI User Guide.

2.11 Accessing the MMG application from Analytical Applications Infrastructure

You can now access the MMG application and its features from AAI.

Prerequisite

- To access the MMG application from AAI, log in to My Oracle Support, and search for **33603448** patch under the **Patches & Updates** tab, download and apply it.
- A valid MMG setup should be up and running

To access the MMG application, perform the following steps:

1. In the AAI configuration schema, update the value for `MMG_UI_URL`, and `MMG_SERVICE_URL` in the `nextgenemf_config` table with the values in the MMG installation.

   Example:

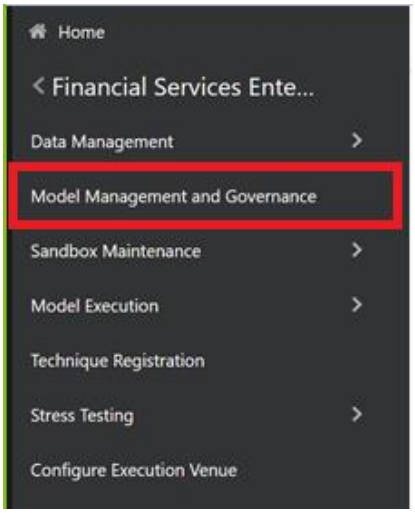
   - `MMG_UI_URL:http(s)://<FQDN/IP>:<mmg_ui_port>/<mmg_context_name>/home`

- `MMG_SERVICE_URL :<FQDN/IP>:<mmg_be_port>/<mmg_context_name>`

> **NOTE** If the target MMG is using secured connection, import the target server certificate to the AAI application java keystore and restart the services.

2. You must map the **MMGACCR** role to the required user. In addition, **MMGACCGRP** group is preseeded with the above role.
3. You must map the `OFS_MMG_HOME` Menu ID to the desired Parent ID in the `AAI_MENU_TREE` table in the AAI Configuration Schema.
4. Restart the AAI services.

   After successful login to the AAI application, a menu named **Model Management and Governance** is displayed. Based on your menu tree configuration, the left pane is displayed as shown below.



5. When you click on Model Management and Governance, the MMG Login page is displayed, enter the credentials to access the MMG application.

3 Using Workspaces

Workspace management allows you to create and manage workspaces in the Financial Services Model Management and Governance application.

Workspace can be in the production environment (deployed), or they can exist in a separate instance on their own (on local for testing purposes) with a copy of data that comes from the desired data source (production or external data source).

You can view the following details on Workspace Summary page:

- Number of Sandbox Workspaces
- Number of Production Workspaces

3.1 Accessing the Workspace Summary Page

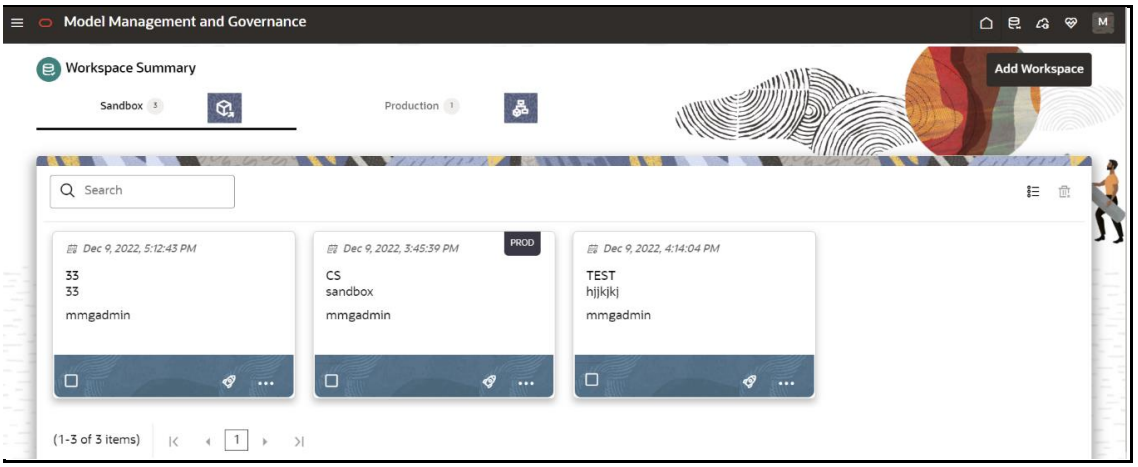
The Workspace Summary page gives access to the various workspace functions such as create and delete.

> **NOTE** Users must be mapped to the Workspace Admin Group to access the Workspace Summary page.

To access the Workspace Summary page, follow these steps:

After login to **Financial Services Model Management and Governance** application, the **Workspace Summary** page is displayed. The page displays workspace records in a table or card view. You can launch a workspace using the launch option or double clicking on it.

Figure 5: Workspace Summary Page



The following table provides descriptions for the fields and icons on the **Workspace Summary** page.

Field or Icon	Description
Search	The field to search for Workspace. Enter specific terms in the field for which you want to search, and press Enter on the keyboard to display the results. You can search a Workspace using Workspace Code, Owner, Creation Date, or Workspace Type fields. The search list shows the Workspace details with Workspace Code, Owner, Creation Date and time, and Type.
Card and Grid View	Click the **Card View**  or **Grid View**  on the top-right corner of this pane to view the workspace as a block or a table, respectively.
Delete	Click **Delete**  to delete the Workspaces.
Workspace Code	The code of the Workspace.
Description	The description for the Workspace.
Owner	The owner of the Workspace.
Creation Date	The date and time on which the Workspace was created.
Data Source Type	Shows the type of the Data Source.
Add Workspace	Click **Add Workspace** to create a new Workspace. See the Create a Workspace section for more information.
Launch Workspace	Click **Launch**  next to corresponding workspace to display the **Workspace Dashboard** with application configuration and model creation menu. See the Launch a Workspace section for more information.
View Workspace	Click Action next to corresponding Workspace and select **View**  to view the workspace with dataset data. See the View a Workspace section for more information.
Populate Workspace	Click Action next to corresponding Workspace and select **Populate**  to populate the workspace with dataset data. See the Populate a Workspace section for more information.
Edit Workspace	Click Action next to corresponding Workspace and select **Edit**  to edit the Workspace.

Field or Icon	Description
Delete Workspace	Click Action ••• next to corresponding Workspace and select **Delete** 🗑 to delete the Workspace. See the Delete a Workspace section for more information.
Download Workspace	Click Action ••• next to corresponding Workspace and select **Download** ⬇ to download the Workspace. See the Download a Workspace section for more information. The file is downloaded in .cfg format.

3.2 Create a Workspace

The Workspace creation requires entry of the source of dataset, validation, and deployment. Besides, the OFS MMG application may require users of different function groups to create and approve a Workspace. In other words, a user associated with the modeler function group creates a Workspace and the approval and deployment are done by a user associated with the modeling administrator function group. See the Mapping User Groups section for more information.

UGDOMMAP function should be mapped to the user performing sandbox creation operation. Otherwise, the create operation will fail.

To create a Workspace, follow these steps:

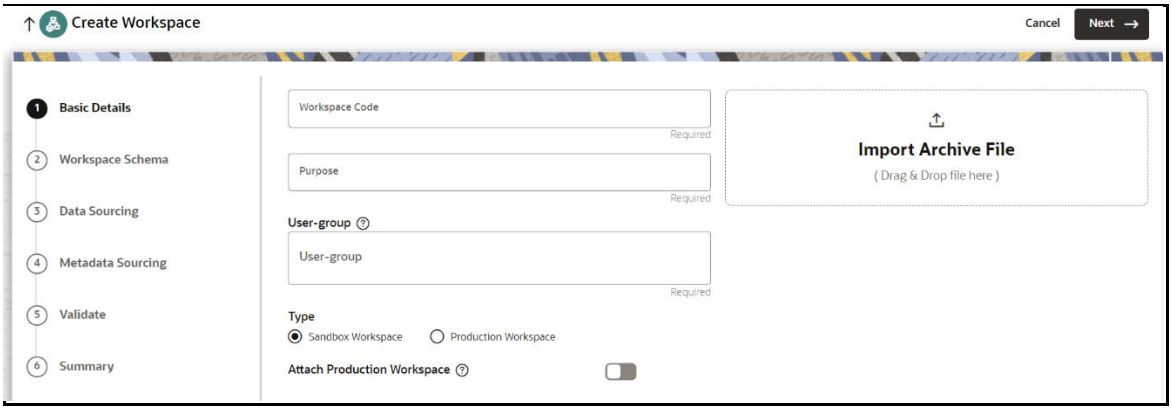
1. Navigate to **Workspace Summary** page.

   The page displays workspace records in a table.

2. Click **Add Workspace**.

   The **Create Workspace** page is displayed.

Figure 6: Create Workspace



The window displays a progress indicator at the left that indicates the active window where you are entering details. Click **Previous** ← Previous to go back a step and click **Next** Next → to go to the next step.

The following steps show the various phases from workspace creation to deployment:

a. Basic Details

b. Workspace Schema

c. Data Sourcing

d. Metadata Sourcing

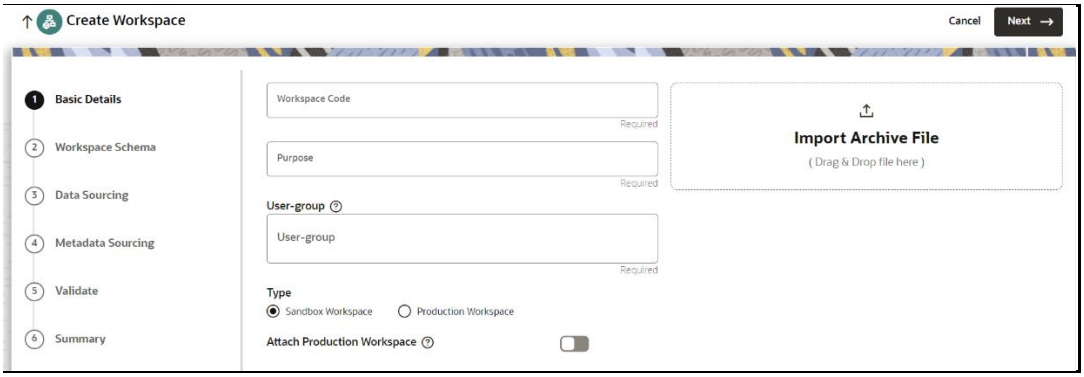
e. Validate

f. Summary

3.2.1 Configure Basic Details

Enter basic configuration details in this window.

Figure 7: Basic Details



To configure the basic details for the workspace, follow these steps:

1. Enter the required details in the **Basic Details** pane as shown in the following table.

Field	Description
Workspace Code	Enter the code of the workspace. This field is limited to 20 characters.
Purpose	Enter the purpose of the creation of the Workspace.
User-group	Click on this field to display a list of User-group values. Select the required value. For Example: Modeling Approver, Modeling Reviewer, and so on.

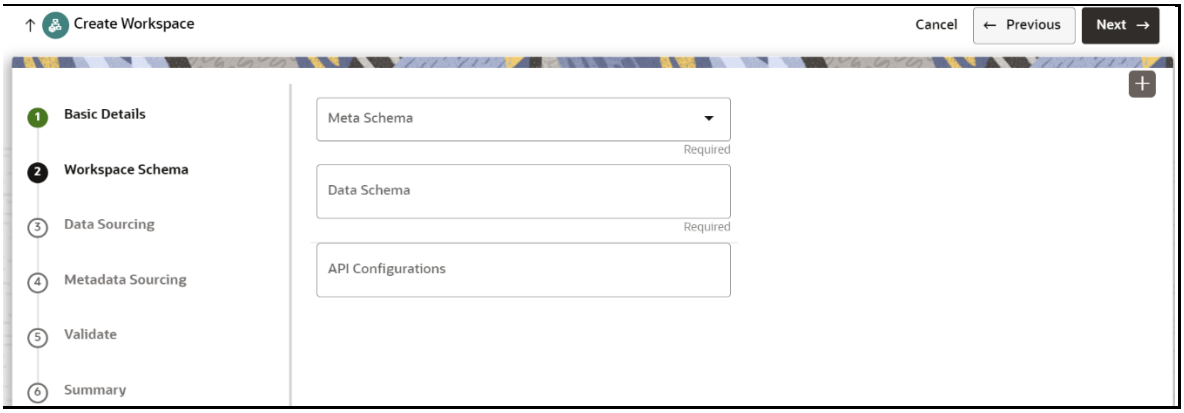
Field	Description
Type	Select the type of Workspace as Sandbox or Production.
Attach Production Workspace	Move the toggle switch to the right to enable this option. Enabling it attaches the production schema, which selects the Workspace being created for automatic model promotion. Based on the selection of the Source Workspace, the model is promoted to the environment.
Import archive file	Enter the archived file to import for basic details. If you use this feature, the other fields described in the preceding rows are auto populated. Click on the box to open the file selector dialog and select the required configuration file or drag the file from its directory and drop it in the box.

2. Click Next → to go to the next step.

3.2.2 Configure Workspace Schema

Select the schema operation and enter connection details.

Figure 8: Workspace Schema



To configure the Workspace schema, follow these steps:

1. Select one from the options from the **Meta Schema** drop-down list.
2. You can upload a new data model with the details you provide in the Data Sourcing drop-down list. You can select an existing Schema or add a new Schema.
   - Meta Source: Every workspace has a schema to store the meta information such as batch ID and so on. This is used for saving all the definitions and metadata related to models.
   - Data Source: This is the actual data used for model building. You can use multiple data schemas for one Meta Source.

To add a new data source, follow these steps:

3. Click Create Data Source  icon.

   The **Create Data Source** window is displayed.

Figure 9: Create Data Source with Oracle Database

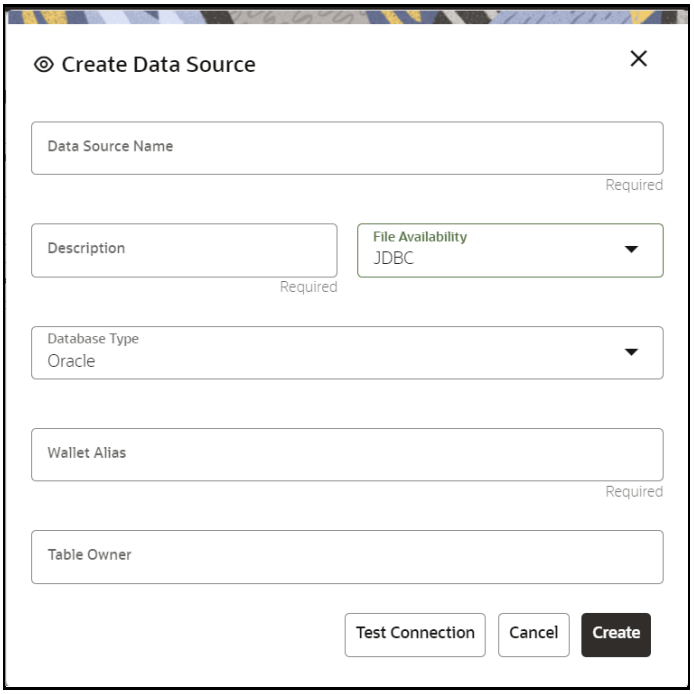
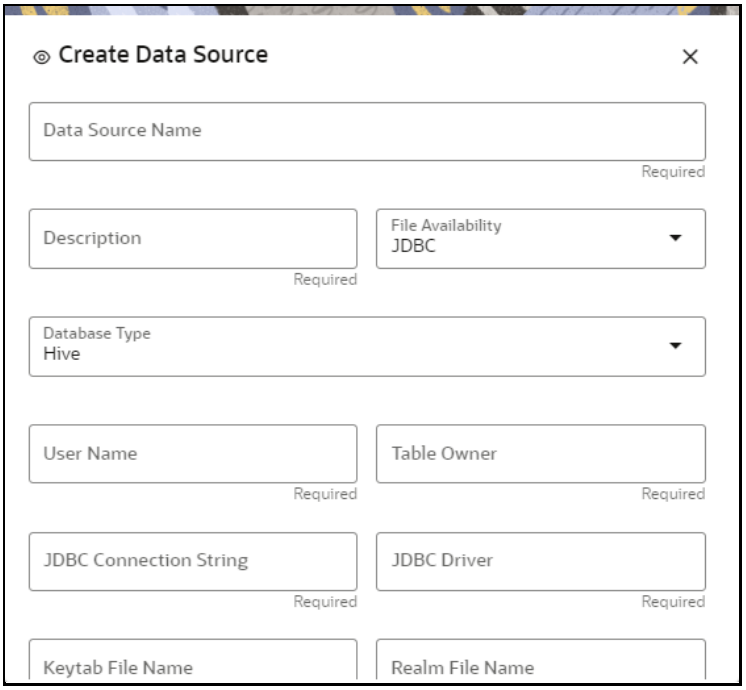


Figure 10: Create Data Source with Hive Database



4. Enter the required details as shown in the following table.

Field	Description
Database Source Name	Enter the connection URL to the database for the data schema.
Description	Enter the description of database connection.
Type	Enter the type of the database connection.
Database Type	Select the Database Type as Oracle or Hive. Note: Selected tables during Hive sourcing should be preexisting in the RDBMS data schema before the workspace population. If you select Database Type as Oracle (see Figure 9), then following additional fields are displayed to enter details: • Table Owner: Enter the Oracle Database name. If you select Database Type as Hive (see Figure 10), then following additional fields are displayed to enter details: • User Name: User Name / Principal is used for Kerberos authentication. • Table Owner: Enter the Hive schema. • JDBC Connection String: Enter the JDBC Connection String. • JDBC Driver: Supports org.apache.hive.jdbc.HiveDriver and com.cloudera.hive.jdbc4.HS2Driver • Keytab File Name: Enter the Name of the keytab file present in conf directory. • Realm File Name: Enter the Name of the configuration file present in conf directory. Note: Schema population for Hive as target is not supported. Note: This is applicable only for Sandbox Workspace. For more information on Setting the Environment, see the Oracle Financial Services Model Management and Governance Installation Guide.
Wallet Alias	Enter the Wallet Alias. This value should be same as configured using Oracle Wallet. For more information, see the Oracle Financial Services Model Management and Governance Installation Guide.

5. Click Create on Create Data Source window.
6. Select the API Configurations.
7. Click 

to go to the next step.

   OR

   Click **Skip** to skip the step.

> **NOTE** Click **Test Connection** to check the connection. A success message is displayed.

3.2.3 Configure Data Sourcing

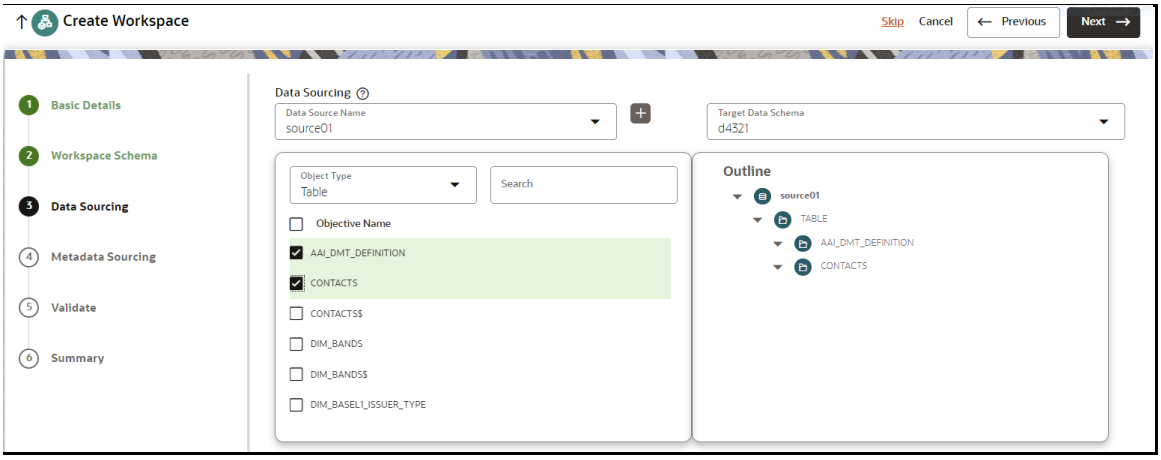
The schema type selected in the previous step requires the definition of database objects to be used for model creation. Data sourcing step of Workspace provisioning allows to select tables from Hive based data sources from which data has to pulled into the Oracle based Workspace data schemas. However, unlike the data sourcing from RDBMS data sources, the tables will not get physicalized in the target schema and hence it is expected that the tables with compatible structures are already present in target RDBMS schema.

In case any of the selected tables are not present in the target schema, those tables are included in the failed objects count in workspace provisioning summary.

This window shows the different icons for Oracle and Hive data sources.

Enter the details in this window.

Figure 11: Data Sourcing - External Data Source



To configure Data Sourcing, follow these steps:

1. You can select Data Source from Data Source Name drop-down list or create a new Data Source. To create a new Data Source, see the Configure Workspace Schema section.
2. Select the Target Data Schema. You can select multiple Data Sources for a Target Data Schema.
3. For example, if there are D1, D2 and D3 Data Sources, then you can select the tables from all these Data Sources, tables from two Data Sources, tables from one Data Source, or as required. Here, multiple combination of tables are possible with Data Source and Target Data Source.
4. If two Data Sources are having same tables (from different Data Sources), then the columns from the first selected table will be used. For example,
5. If Table A has columns C1, C2, C3 and Table B has columns C1, C2, and C4,then the data from the first table will be used.

6. During the data population, only columns C1 and C2 will be used and those will be marked in Green color.
7. Select the type of objects to be displayed in the pane that follows the drop-down list. The Object Type drop-down list will be enabled after selecting the Data Source from Data Source Name drop-down list. The following are the options in the drop-down list:
   - Table
   - View
   - Synonym
8. Click Next → to go to the next step.

   OR

   Click **Skip** to skip the step.

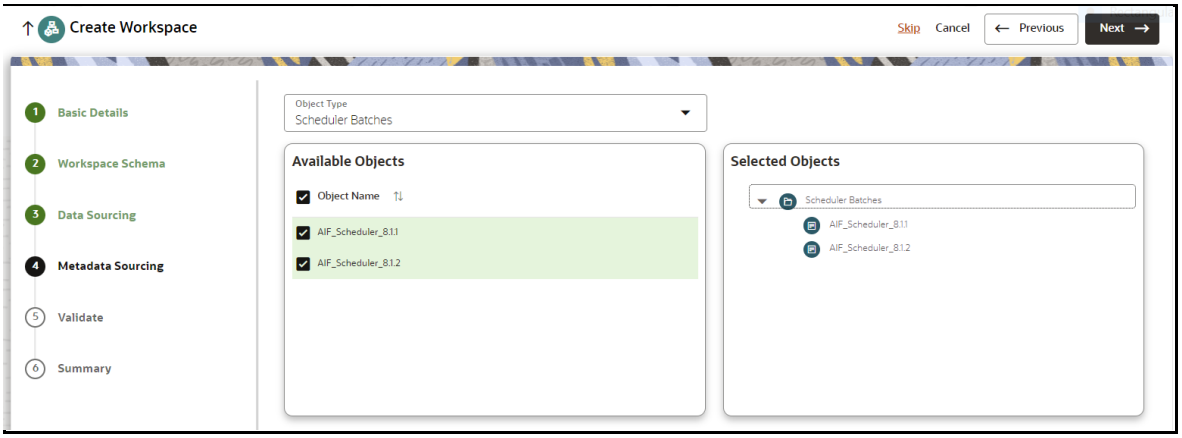
3.2.4 Configure Metadata Sourcing

The database objects selected in the previous step can be added with metadata for selected objects. Metadata Sourcing is a stage during Workspace provisioning to allow seeding of metadata like scheduler batches at the time of workspace provisioning.

Also, by default there will be seeded metadata. However, if you wish to seed the metadata, navigate to <installed path>/ftpshare/mmg/seeded/batches and drag and drop the metadata in SQL format.

> **NOTE** This step is optional.

Figure 12: Metadata Sourcing



To configure Metadata Sourcing, follow these steps:

1. Select the required schema from the **Object Type (Optional)** drop-down list.

   The **Available Objects** are displayed in **Selected Objects** pane.

2. Click Next → to go to the next step.

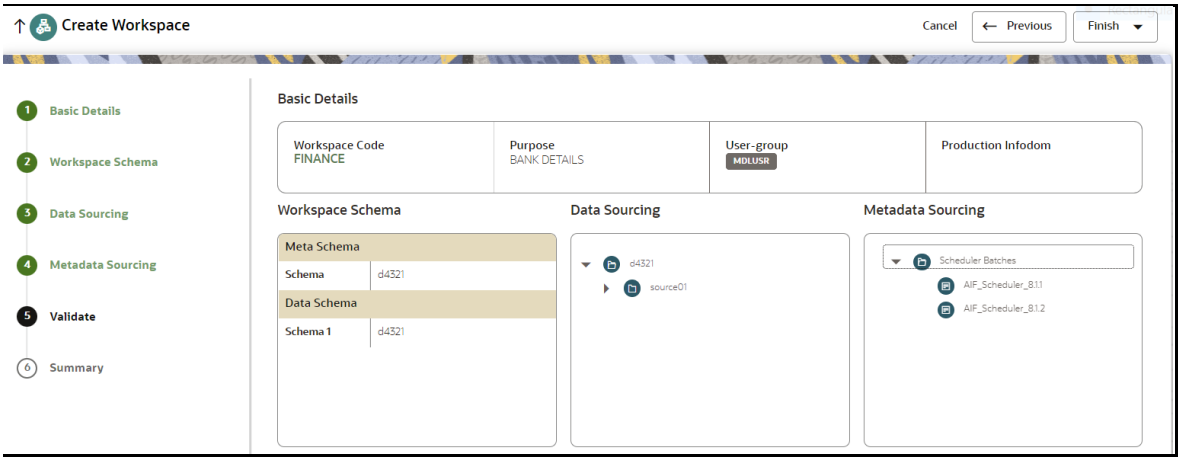
   OR

   Click **Skip** to skip the step.

3.2.5 Validate Workspace

The **Validate** pane displays a preview of the configuration values entered in the previous panes.

Figure 13: Validate Workspace



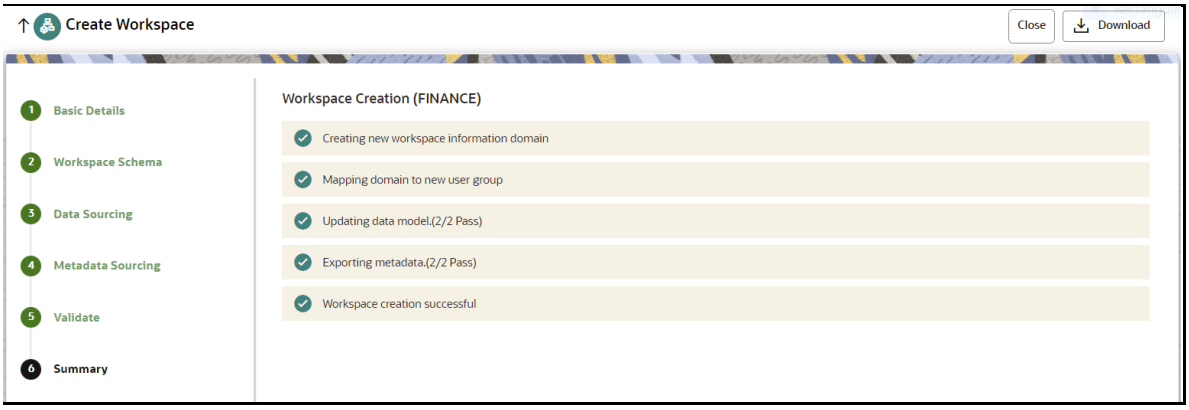
To validate the Workspace and deploy, follow these steps:

1. Review the details in the **Validate** pane. You can edit the Workspace by clicking on Edit icon .
2. Click **Finish** to creation of the Workspace using **Physicalize Sandbox** option or **Download Configurable Archive**.
   - When you click **Download Configurable Archive**, it exports the metadata information of the workspace in .zip format which can be used later using the Import option.
   - When you click **Physicalize Sandbox**, it creates actual workspace.

3.2.6 Display Summary

The **Summary** pane displays the status of the workspace creation.

Figure 14: Workspace Creation Summary



- Click 

to download the deployment report.

3.3 Launch a Workspace

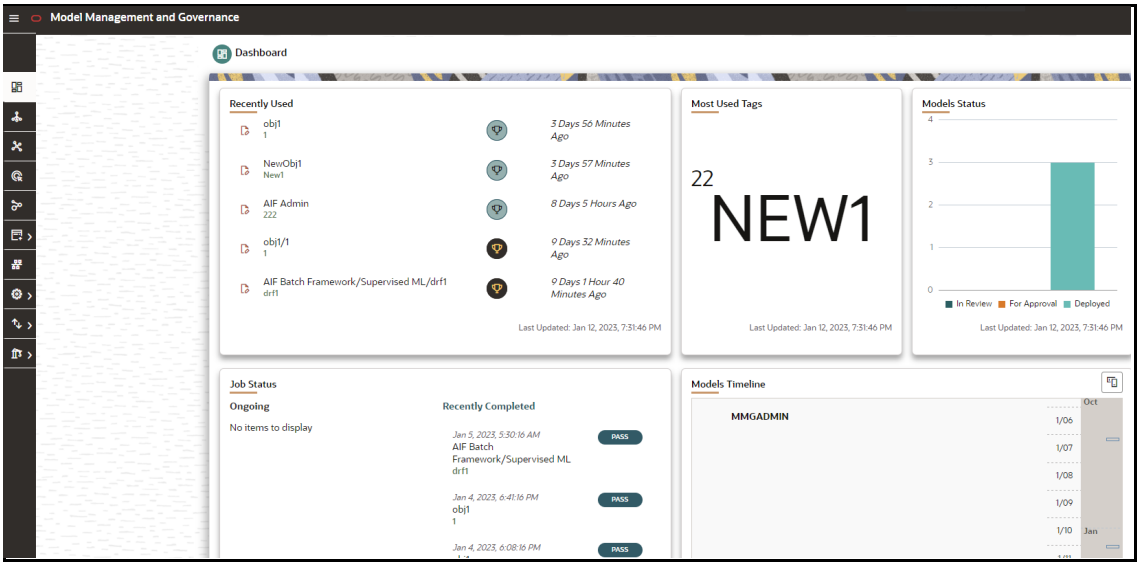
The launching of the Workspace displays the **Workspace** window. The workspace displays a menu for Models and an application configuration and model creation submenu as shown in the following list:

- Dashboard
- Dataset
- Model Pipelines
- Model Actions
- Graphs
- Scheduler Service
- Audit Trail
- Data Studio Options
- Object Migration
- Model Training

1. Click Launch  next to corresponding Workspace.

   The Workspace Dashboard is displayed.

Figure 15: Workspace Dashboard



3.4 View a Workspace

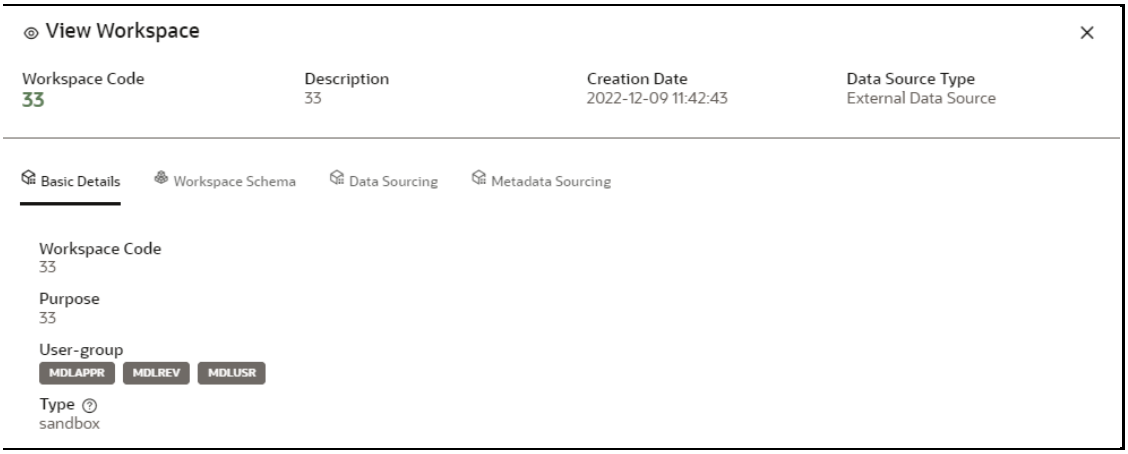
To view a workspace, follow these steps:

1. Navigate to **Workspace Management** to display the **Workspace Summary** page. The page displays workspace records in a table.

2. Click **Action** next to corresponding workspace and select **View**  to view the workspace.

   The Basic Details of the Workspace is displayed.

Figure 16: View Workspace Window



3. Navigate to the Workspace Schema, Data Sourcing, and Metadata Sourcing to view the respective details.

3.5 Populate a Workspace

The workspace is populated with data from the datasets in External sources.

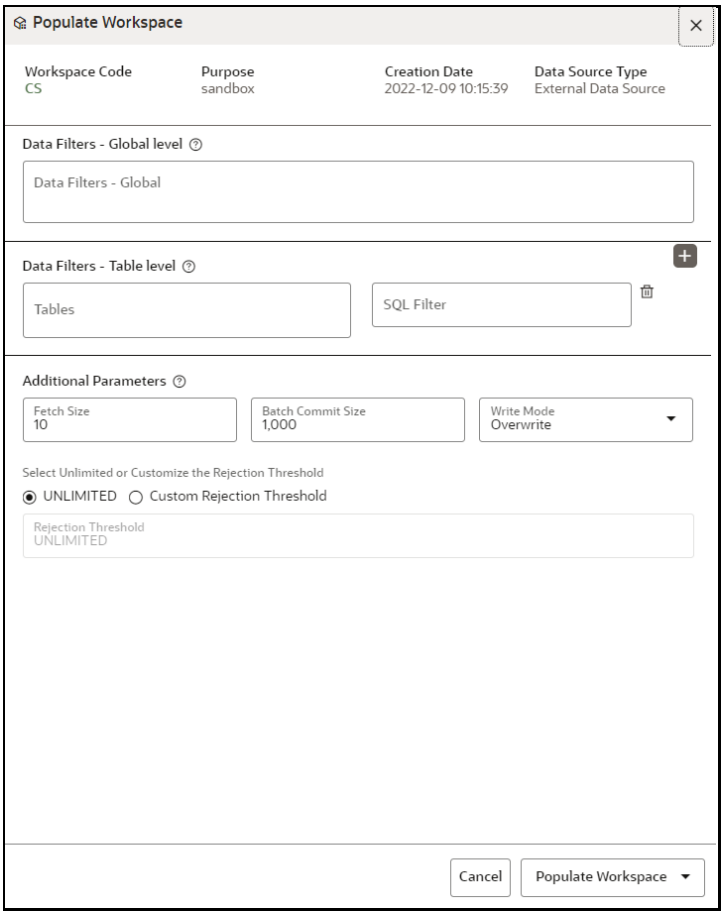
To populate the Workspace, follow these steps:

1. Navigate to the Workspace **Summary** page.

   The page displays Workspace records in a table.

2. Click Action ••• next to corresponding Workspace and select **Populate** to populate the Workspace with data from a dataset in the **Populate Workspace** window.

Figure 17: Populate Workspace Window



The following table provides descriptions for the fields in the **Populate Workspace** window.

Field	Description
Workspace Code	The code of the Workspace.
Purpose	The description for the Workspace.
Creation Date	The date and time on which the Workspace was created.
Data Source Type	The source of data. The value can be the OFSAA Data Schema or an external data source.
Data Filter - Global	Enter the data filter that needs to be applied on all the tables selected for data sourcing. For example: If MISDATE is equal to Today, then it is applied to all tables (wherever it is available) for selected Data Sources during population. If this field is not found (MISDATE) in the tables, it is not updated.
Data Filter - Table level	Provide the data filters individually on the tables here. **NOTE:** You can provide multiple table names for the same SQL filter. For example, there are two tables called Student and Employee in the target data source, and below filters are applied MISDATE as Today for Student and Employee tables ID as 1 for Student table Then, Student table will be populated with MISDATE and ID filters and Employee table will be populated with only MISDATE filter. Global Filters will not be applicable for those tables on which filters have been applied individually. If the same table name is provided in more than one rows here, then filter condition is generated as a conjunction of all the provided filters.
Fetch Size	Enter the Fetch size of JDBC properties for data upload
Batch Commit Size	Enter the Batch Commit size of JDBC properties for data upload
Write Mode	You can either overwrite the existing data (truncate and insert) or to append to the existing data. You can choose to either overwrite the data or append to the existing data.
Rejection Threshold	Following two options are available: • Custom Rejection Threshold Enter the maximum of number of inserts that may fail for any of the selected tables. You can provide the maximum number of inserts that can fail while loading data to a given table from all the sources. In case of threshold breach, all the inserts into the particular target schema will be rolled back. However, it will continue with populating the next target schema. • Unlimited Here, all the errors will be ignored during the data population.

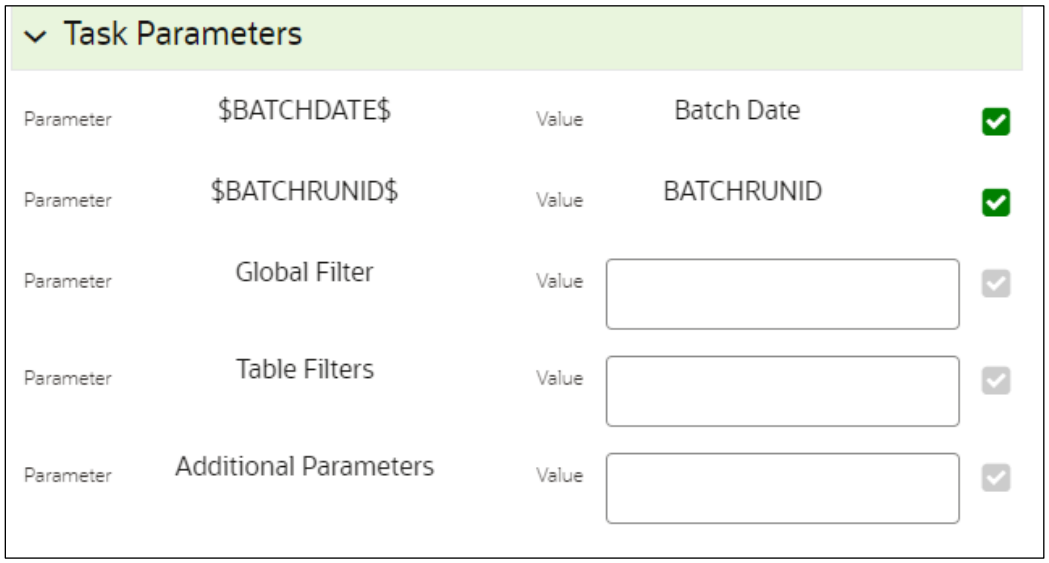
3. Click **Populate Workspace** to start the process.

   Here, you can create the batch using Create Batch, or create and execute using Create and Execute Batch option. On selecting either of these options, a workspace population task gets added to the batch.

- When you select Create and Execute Batch option, it allows you to create batch and triggers the batch as well.
- When you select 'Create Batch' option, it allows you to prepare the batch and then execute or schedule the batch at a later time through Scheduler Service window.

The Workspace population task execution can be tracked in the 'Monitor Batch' window in the Scheduler Service.

Figure 18: Task Parameters



4. Enter the following parameters for workspace population.

> **NOTE** This step is optional.

- **Additional Parameters:** Enter the Additional Parameters in following format:

  {"fetch_size" :10, "batch_commit_size" :1000,
  "rejection_threshold" :"UNLIMITED", "write_mode" :"OVERWRITE"}

- **Global Filter**

  Provided input will be applied as a data filter on all the tables selected for data sourcing.

- **Table Filter**

  You can provide data filters individually on the tables here. You must provide multiple table names for the same SQL filter. Global Filters will not be applicable for those tables on which filters have been applied individually. In case the same table name is provided in more than one rows here, the filter condition will be generated as a conjunction of all the provided filters.

Enter the Table filters in following format:

[{"id":1,"filter":"<filter condition>","tables":["TABLE1", "TABLE2"]}, {"id":2,"filter":"<filter condition>","tables":["TABLE2"]}]

NOTE You can run workspace population for a given workspace any number of times. New tables may be added to the definition. Any new table added to the definition, that is not present in the target schema will be physicalized on update of the workspace. Also, user can add new sources if required.

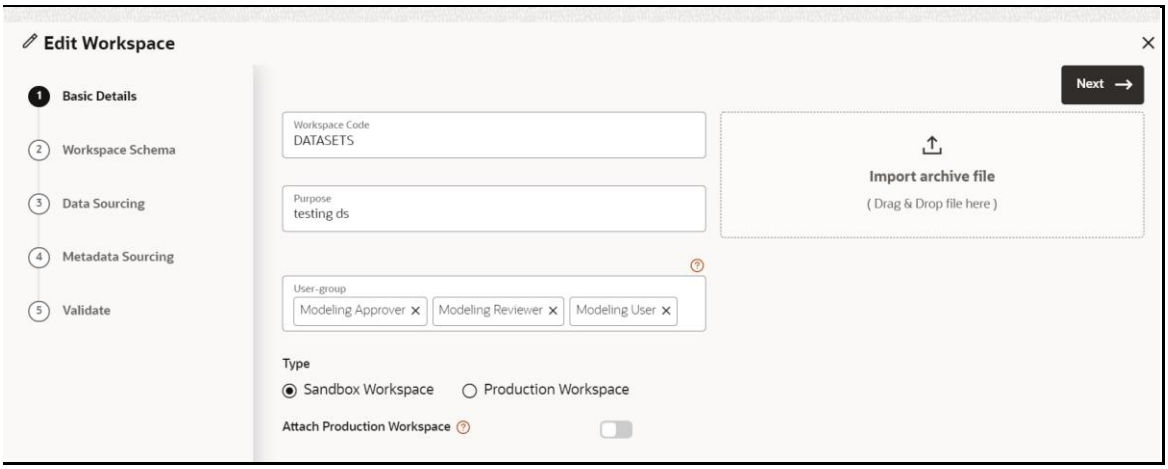
Any table that is deselected from the data sourcing definition will **NOT** be dropped.

3.6 Edit a Workspace

To edit a workspace, follow these steps:

1. Navigate to **Workspace Management** to display the **Workspace Summary** page. The page displays Workspace records in a table.
2. Click Action ••• next to corresponding workspace and select **Edit** to edit the workspace.

Figure 19: Edit Workspace Window



NOTE You can modify the Workspace Type from Sandbox to Production or vice versa.

For more details on the Basic Details fields, see Configure Basic Details section.

3. Click **Save**.

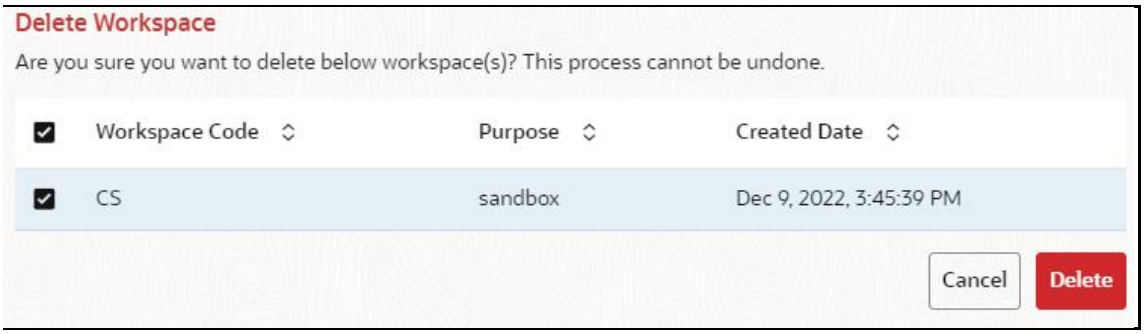
3.7 Delete a Workspace

> **NOTE** When you delete a Workspace, all the underlying objects from the associated tables are deleted.

To delete a Workspace, follow these steps:

1. Navigate to **Workspace Management** to display the **Workspace Summary** page. The page displays workspace records in a table.
2. Click Action ••• next to corresponding Workspace and select **Delete** to delete the workspace.

Figure 20: Delete Workspace Window



The following table provides descriptions for the fields in the **Delete Workspace** window.

Field	Description
Workspace Code	The code of the workspace.
Purpose	The description for the workspace.
Created Date	The date on which the workspace was created.

3. Click **Delete**.
4. Click **OK** on the confirmation dialog box to confirm or click **Cancel** to cancel.

> **NOTE** You must de-link the Sandbox workspaces from Production Workspace before deleting the Production Workspace

3.8 Download a Workspace

Downloading a workspace allows you to export the workspace metadata definition (without underlying objects such as Models, Pipeline, Graphs, and so on) in a zip format. Further, the same can be used to re-create the workspace.

To download a workspace, follow these steps:

1. Navigate to **Workspace Management** to display the **Workspace Summary** page. The page displays Workspace records in a table.
2. Click Action ••• next to corresponding workspace and select **Download** to download the workspace.

   OR

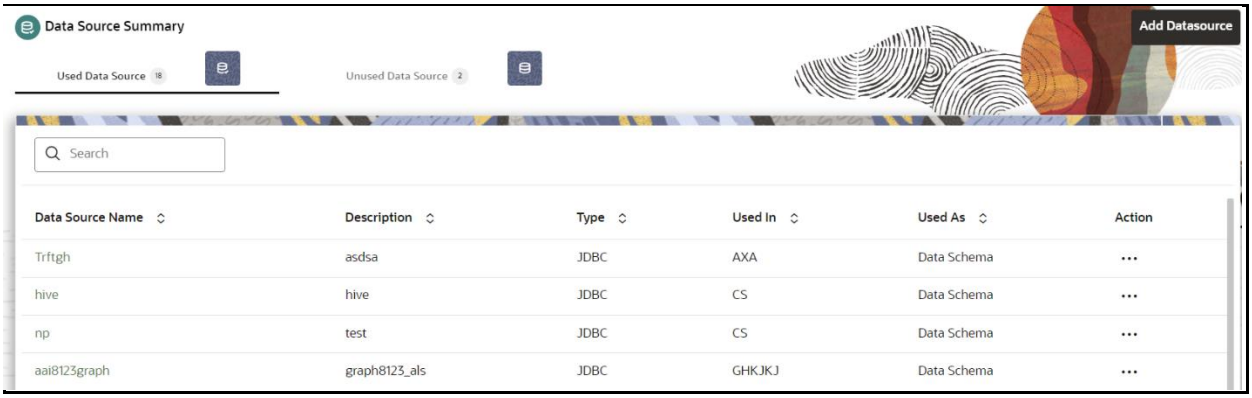
   Navigate to the Summary screen during the creation of a Workspace and click Finish and select **Download Configuration Archive** option.

> **NOTE** The file is downloaded in a zip format.

3.9 Managing Data Sources

This feature allows you to manage the Data Schemas registered with MMG. The Data Source Summary window shows the list of Data schemas registered with MMG. These Data schemas can be used either for workspace or for sourcing data. Click Data Source Summary to navigate to Data Source Summary window.

This window also allows you to manage these registered external sources.



The Data Source Summary is divided into two sections: Used Data Source and Unused Data Source.

To view the Data Source details., Click **Action** icon next to corresponding Workspace and select **View**

- **Used Data Source:**

  This shows the list of Data Sources registered with any workspace. Here, you can only view the Data Source details. The count of Used Data Sources also displayed at the top of Data Source Summary page.

- **Unused Data Source:**

  This shows the list of Data Sources those are not registered with any workspace. Here, you can only view, edit, or delete the Data Source details. The count of Unused Data Sources also displayed at the top of Data Source Summary page.

To add a Data Source from Workspace Summary window, follow these steps:

1. Navigate to **Workspace Summary** window.
2. Click Data Source Summary to navigate to **Data Source Summary** window and click Add Datasource.

   The Add Data Source window is displayed.

To add a Data Source from Data Source Summary window, follow these steps:

1. Navigate to **Data Source Summary** window.
2. Click **Add Datasource** icon Add Datasource.

   For more information, see the Create a Workspace section.

4 Dataset

Dataset allows you to manage the entire operation related to dataset. You can perform the following two things using the Dataset window:

- Define a metadata on how you want to create a dataset
- In addition, a mechanism where you can actually take a snapshot of real time data and store it. So, it can be used later in the pipeline.

This is similar to T2Ts, where you can select data from Data Source, such as file, table or another dataset and so on. The data can be imported from one column of one table, and another column from another table, or any file. After extracting the data from tables or files, provide the name to Target Dataset.

The Dataset is a trail-based UI that allows you to configure the dataset details.

4.1 Accessing the Dataset Summary Page

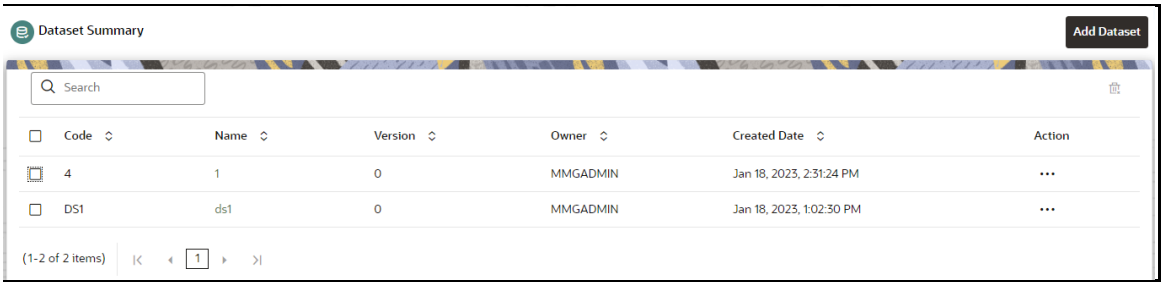
The Dataset Summary page gives access to the various Dataset functions such as create, view, and delete.

To access the Dataset Summary page, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. In the LHS menu, click **Dataset**  to display the **Dataset Summary** window.

   This window displays the dataset records in a table.

Figure 21: Dataset Summary Page



The following table provides descriptions for the fields and icons on the **Dataset Summary** page.

Field or Icon	Description
Search	The field to search for Dataset. Enter specific terms in the field for which you want to search, and press Enter on the keyboard to display the results.
Code	The code of the dataset.
Name	The name of the dataset.
Description	The description for the Workspace
Version	Version of dataset
Created Date	The date on which the Dataset was created.
Owner	The owner of the dataset.
Add Dataset	Click Add Dataset to create a new Dataset.
Delete	Click Delete to delete multiple Datasets.
Action	Click the three dots to perform View/Edit/Delete/ Profile functions on selected dataset.

4.2 Create a Dataset

The Dataset creation requires entry of the source of dataset and validation. These datasets are required for schema creation.

4.2.1 Creating a Dataset

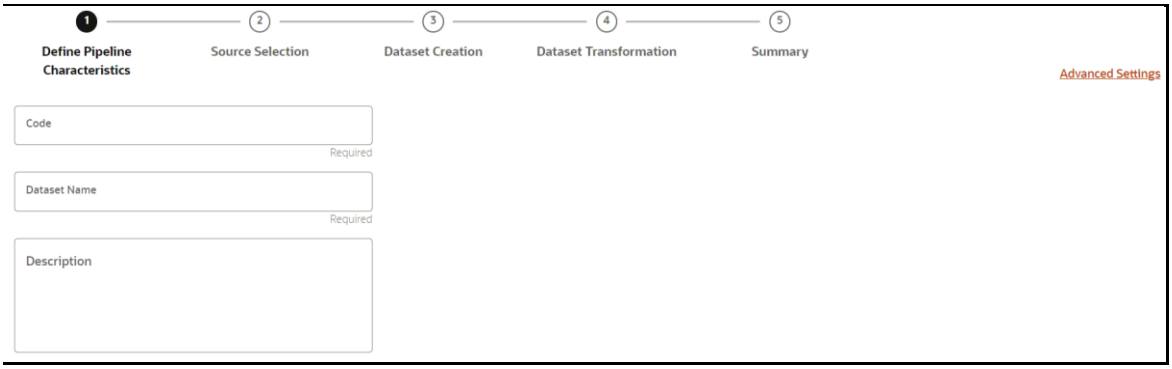
To create a Dataset, follow these steps:

1. Navigate to Dataset Summary page to display the **Dataset Summary** page.

   This page displays the dataset records in a table.

2. Click **Add** to create a Dataset in the Dataset **Summary** window.

Figure 22: Dataset Creation Window



The following steps show the various phases from Dataset creation:

- Define Pipeline Characteristics
- Source Selection
- Dataset Creation
- Dataset Transformation
- Summary

4.2.2 Define Pipeline Characteristics

Enter basic configuration details in this window.

To configure the basic details for the dataset, follow these steps:

1. Enter the required details in the **Define Pipeline Characteristics** window as shown in the following table.

Field	Description
Code	Enter the code of the dataset. This field is limited to 30 alphanumeric characters.
Dataset Name	Enter the name of dataset. This field is limited to 30 alphanumeric characters. Space not exceeding 30 characters. You cannot keep this field blank.
Description	Enter the purpose of the creation of the dataset. This field is limited to 150 alphanumeric characters. Space not exceeding 150 characters.

2. Click **Advanced Settings** to select the Dataframe Library.

   The Advanced Settings window is displayed.

3. Select the Dataframe Library for the Dataset as Pandas (Default) or Modin and click **Close**.
   - **Pandas**: Pandas is an open-source data manipulation library for Python. It provides data structures such as Series (1-dimensional) and DataFrame (2-

dimensional) that allow for easy manipulation and analysis of data. It also provides tools for reading and writing data to various file formats, including CSV, Excel, and SQL databases.

By default, **Pandas** is selected.

- **Modin**: Modin is an open-source library that allows for faster operations on DataFrames using distributed computing which can lead to significant speed improvements, particularly for large datasets or computationally expensive operations.

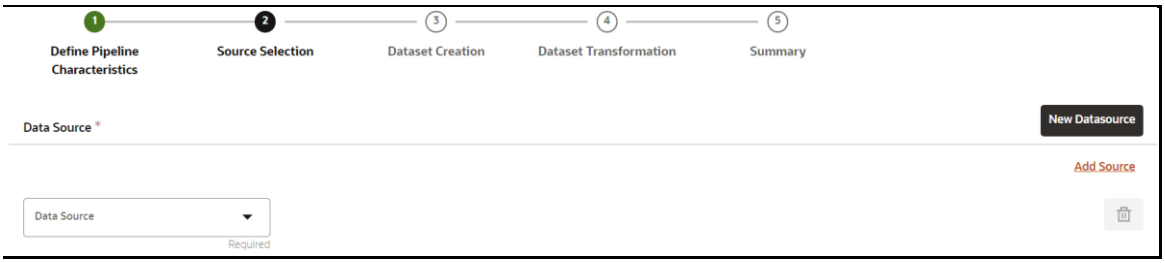
4. Click **Next** to go to the next step.

4.2.3 Source Selection

This section allows you to define the source of data. Here, you can choose the data structures from an existing datasources.

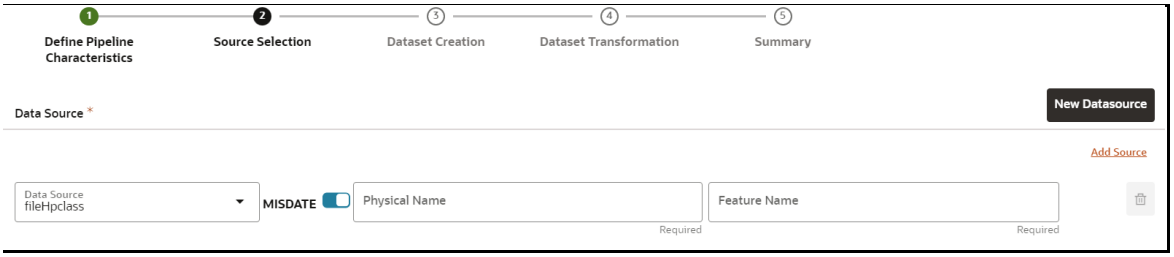
To configure the Source details for the dataset, follow these steps:

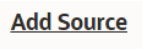
1. Enter the required details in the **Source Selection** pane as shown in the following table.

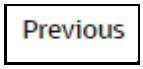


NOTE The MISDATE feature is displayed only when the Data Source field is selected as a **File Type**.

If enabled, you need to add the Physical and Feature name for the Data Source.



Field	Description
Data Source	The Data Source drop-down list shows all the data sources and the data dorms those are tagged with the corresponding workspace. In addition, this lists all the available unused data source, which can be used. A confirmation message is displayed to tag an unused datasource with workspace.  Here, either use the unused or create a new datasource using New Datasource  . When you create a new datasource, it becomes as unused. For more information, see Creating Data Source. You can add more Data Sources using Add More  link. Use Delete  to delete added Data Source.

2. Click  to go to the next step or click  to back to previous step.

4.2.4 Dataset Creation

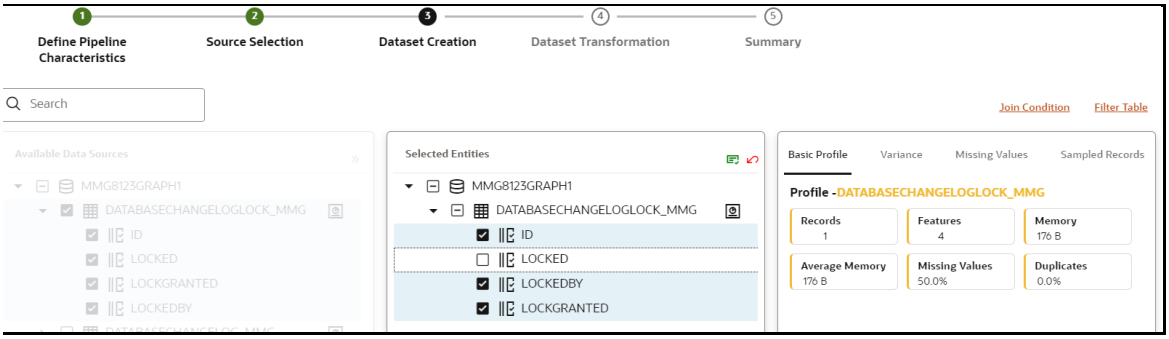
This window allows you to select entity part (for example, column of table) of the datasource. The data source selected in the previous step requires the definition of database objects to be used.

To configure the Dataset Creation, follow these steps:

1. Select the required data source from the **Available Data Sources pane** and click >> to move the selected data to **Selected Entities** pane. The **Available Data Sources** section shows the high-level data sources. You can select data from multiple data source entities. The selected Data Sources are displayed in **Selected Entities** pane.

   You can use  to select or use  to de-select the data in the **Selected**

Entities pane. Also , clicking on **View Profile** icon displays the profile details in the **Basic Profile** tab.



2. For more information on Joins, see Joining Tables section.

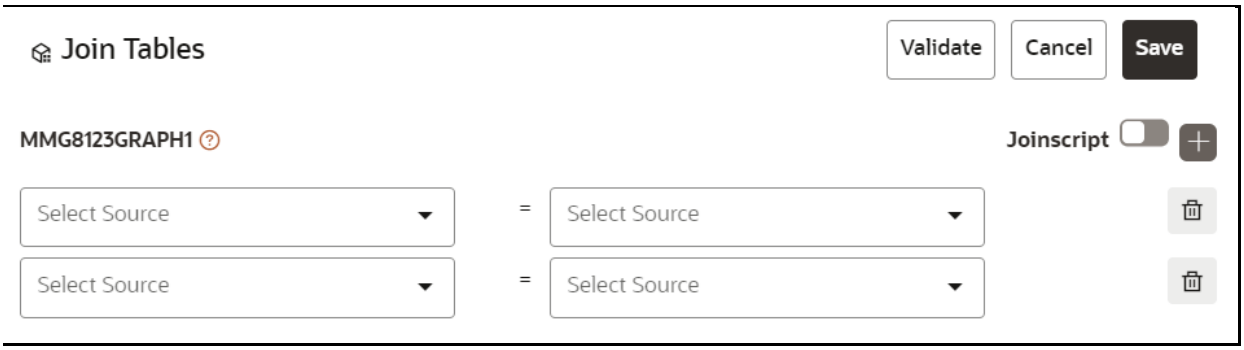
   For more information on Filtering, see Filtering Tables section.

3. Click **Next** to go to the next step or click **Previous** to back to previous step

4.2.4.1 Joining Tables

Use Join Condition to combine the data source details. Each data sources have multiple tables and multiple columns, so you can join them using Join Condition window. Joins can be applied only when user has selected more than 1 table

NOTE: As of now, only INNER JOIN is present.



You can join Tables either using drop-down or using the Joinscript option.

4.2.4.1.1 Using drop-down

Select the Column name of Table from the drop-down list. You can use multiple join conditions by using + icon

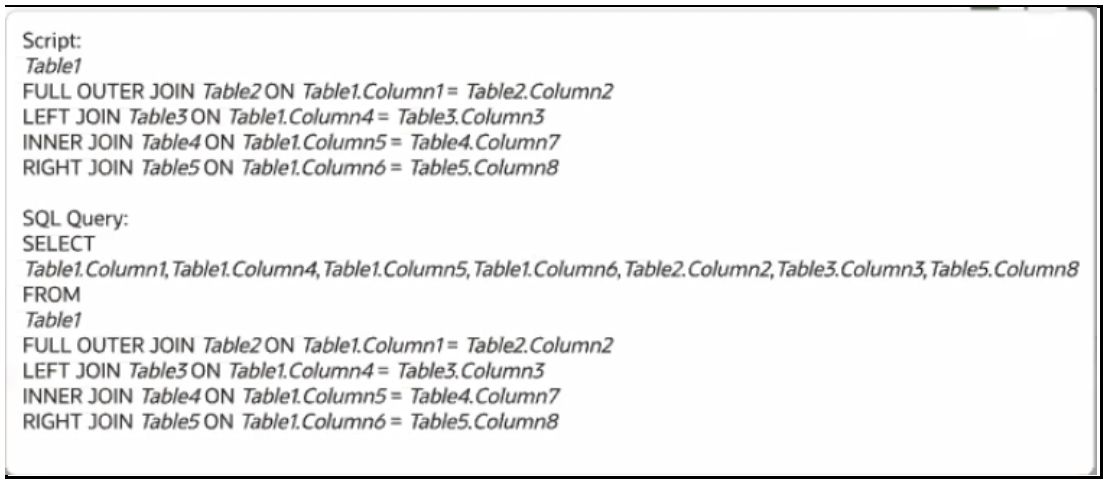
4.2.4.1.2 Using Script

1. To use script, enable 

option.

   The Join Tables window is displayed to add scripts:

You must provide the script in below format:



2. Click **Validate** to check the Join conditions.
3. Click **Save**.

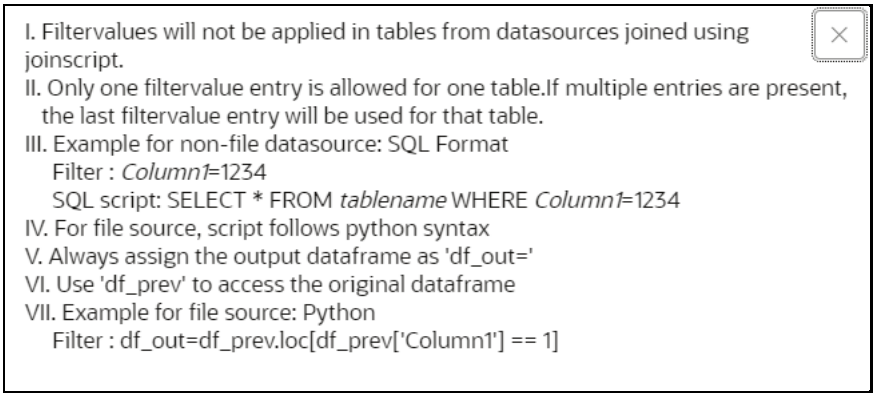
4.2.4.2 Filtering Tables

You can also use the filters in dataframe creation from entity.

1. Click 

to navigate to Filter Tables window.

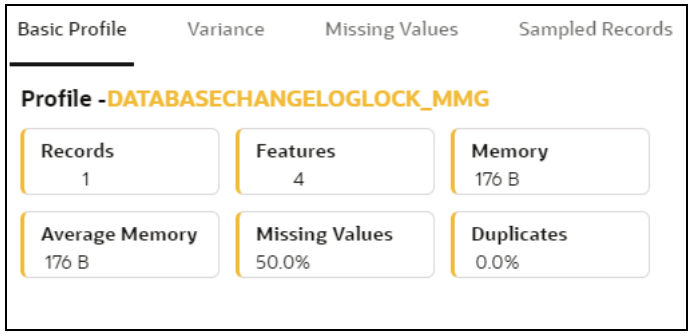
There are certain rules to be followed when adding filters



2. Multiple filters can be added by using  icon.
3. Click **Validate** to check the Filter conditions.
4. Click **Save**.

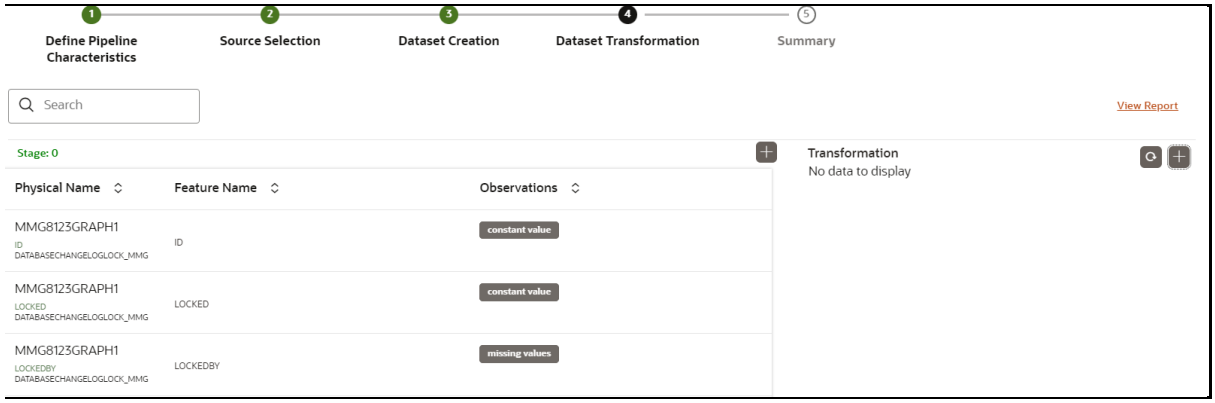
4.2.4.3 Viewing Profile

The Profile section from RHS shows the report based on Selected Objects (complete dataset) section. For example, Missing Values report shows the details of missing data in selected table in Selected Objects section.



4.2.5 Dataset Transformation

This window allows you to transform the data source information. This complete grid displays the Table like structure, which helps you to make the data better using many methods, such as by remove all the missing values, performing scaling and so on.



This window shows the following columns:

- **Physical Name:** Shows the Physical Name of column.
- **Feature Name:** Shows the Physical Name of column. You can edit the Logical Name.
- **Observations:** The Observations field shows the value of column. For example, if any column is missing any value, you can easily identify missing value columns. Use Transformation button to fix these values.
- **Create Feature** icon: To create a new feature/column by doing operations on the existing features.
- **View Report:** To view the complete report of the sampled data
- **Re-Validate** icon**:** To re validate all the transformations.

  You can also revalidate an individual transformation by clicking the More Action icon and select **Re-Validate** option.
- **Add Transformation** icon**:** To add new transformations

The profile button helps you to view profile the final data frame that you have selected. For more information, see the Viewing Profile section.

Here, you can perform following actions

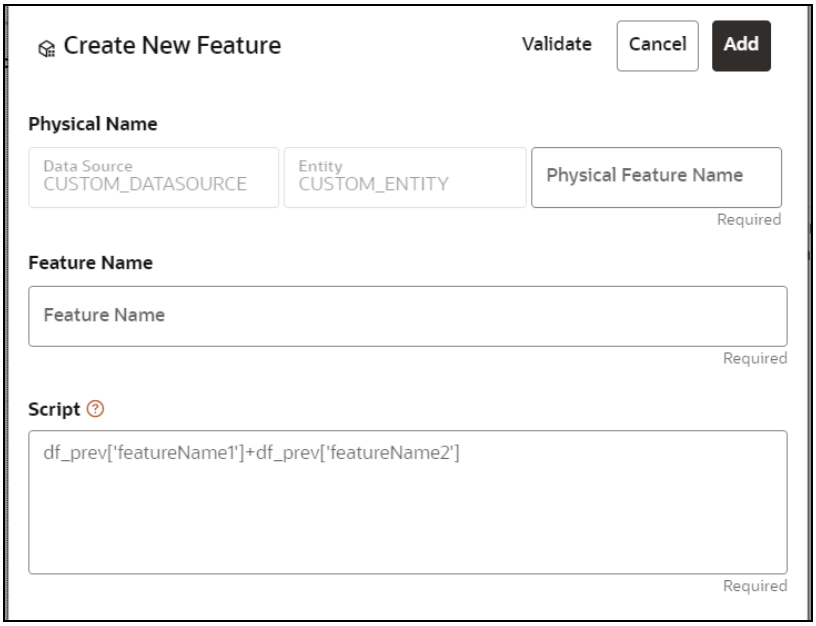
- Re-ordering of transformations
- Insertion of transformation

4.2.5.1 Create New Feature

The Create New Feature window allows you to create a new feature. This is used to add a new column to dataset. This is useful, if you want to add a new column based on derived T2Ts (from data source).

To create a New Feature, follow these steps:

1. Click Create Feature on Dataset Transformation of Dataset UI.



2. Enter the following details:
   - **Physical Feature:** Name of Physical feature
   - **Feature Name:** Logical Name of feature
   - **Script:** Update the script
3. Click Add after validating the feature.

   The new feature will be added at end of LHS section.
4. You can also edit or delete a newly added feature.

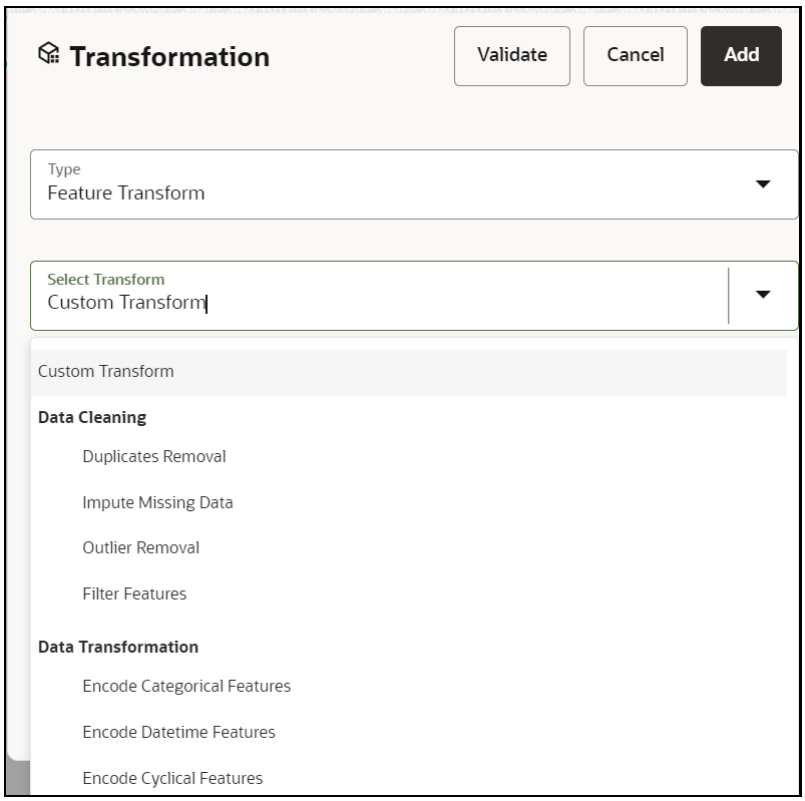
4.2.5.2 Add Data Transformation

To add the data transformation, follow these steps:

1. Click **Add**.

   The Transform window is displayed.
2. Select the Transformation Type.

   Following types are available:
   - **Dataframe Transform:** This is used to transform entire dataframe. For example, if you want to remove all missing values from all the columns of entire sampled data.
   - **Feature Transform:** This is used to transform a particular column of dataframe.



3. **Select Transform:** Here you either enter values using Method and Argument fields, or script.
4. Click **Validate** to validate the details.
5. Click **Add** to add the new transformation.

   The New Transform is created and displayed at RHS section.
6. Click **Finish** to navigate to Dataset Summary window.

   This saves the metadata of dataset.

Example:

If there is “missing value” for one of the columns, then perform following steps to add the transform.

1. Click Add on Dataset Transformation window.
2. Select Type as Feature Transform.
3. Select Transform as Impute Missing Value
4. Select the Physical Name from Physical Name drop-down list.
5. Select the Method and enter parameter in Argument field. The Method is updated on selected Column type. For example, if the selected column type is numerical, then following methods will be available: Simple, Constant, KNN, and Mice

   OR

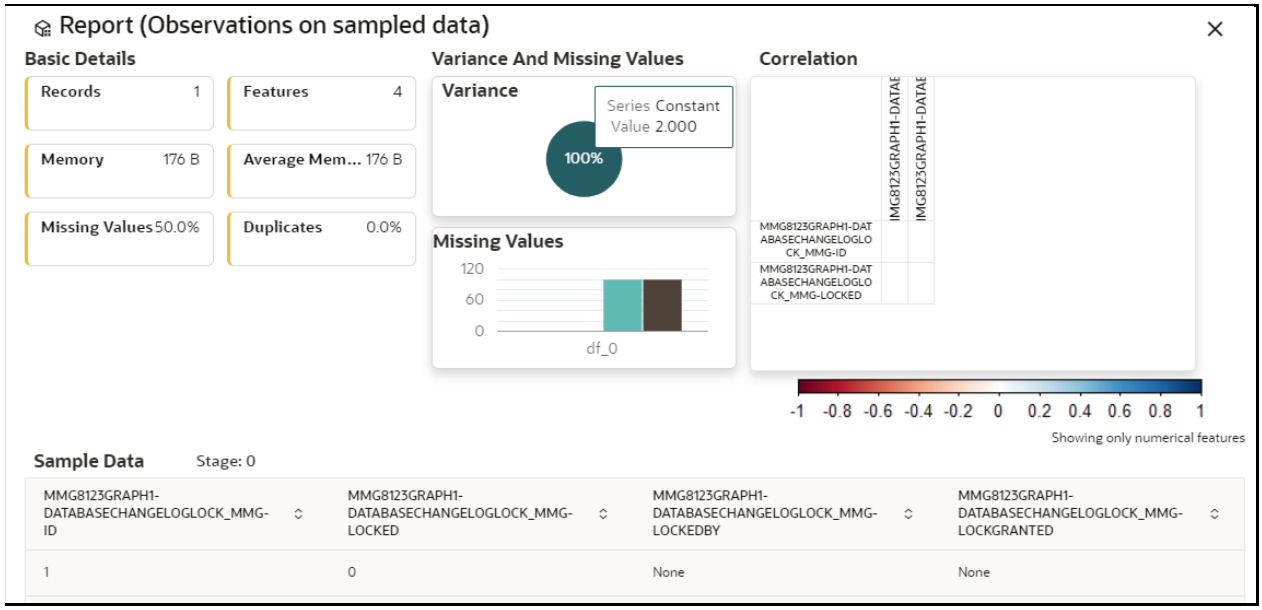
Enter the script

Below are Sample Custom Scripts

- Directly pass input data frame to output: df_out=df_prev
- Drop first row of data frame: df_out=df_prev.drop(0,axis=0) if not df_prev.empty else df_prev
- Drop column from data frame: df_out = df_prev.drop('colname',axis=1)

4.2.5.3 View Reports

This report shows the Observation on Sampled data, profiles, Variance, Correlation Matrix (correlation between columns), Sample Data. This report is based on sampled data.



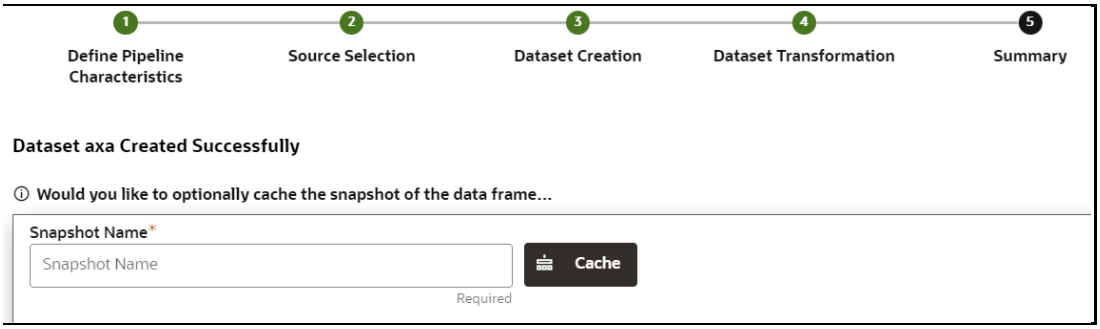
4.2.5.4 Re-ordering of Transformations

You can re-order the transformations using the drag-drop. During the transformation re-order, you can compare the profile of transformations. The transformation order gets adjusted accordingly.

4.2.6 Display Summary

The **Summary** pane displays the status of the dataset creation.

Figure 23: Dataset Creation Summary



A confirmation message is displayed as "Dataset #DATASETNAME# is created successfully."

Enter the **Snapshot Name** having alphanumeric character, underscore and hyphen not exceeding 30 characters and click on Cache. This screen additionally allows users to cache a snapshot of the current state of data. The cached snapshots can be accessed later in Model Pipelines without any recompilation or re-read from data sources.

To use the dataset in model pipeline or data pipeline, the actual data is fetched using the Cache option. For example, to take the data from dataset on As of Date, create the data frame and provide the name to cache. Only Cache pulls the data from dataset.

This helps the things to work faster when you have millions of data, and you want to use intermediate data for use. For example, if you have 1 million data and want to use only 10,000 out of that, then perform the sampling for that 10, 000 entries. This increases the speed of processing, validation of information.

- Once the metadata is created, the original data can be cached. A snapshot of the actual data in the dataset at the current time can be stored referenced by a tag name.
- The location for caching is in the datastudio server location `$DS_HOME/work/ftpshare/mmg/workspace_name/dscode/tag`. The dataset will be saved as a parquet file with name dscode_tag.parquet
- When executing all APIs from notebook, workspace has to be attached.
- Caching can be performed in 2 ways
    - From UI, immediately after saving the metadata

Or the users will have to fetch(create) a new snapshot/dataframe of the dataset using the API 'Fetch New Snapshot of dataset' and manually cache using the 'Caching Data Frame' API.

The following table provides information on the error and the troubleshooting procedure in case of dataset failure.

Error	Troubleshooting procedure
`ModuleNotFoundError: No module named '_bz2'`	Install the package 'libbz2-dev' before building python.
`ModuleNotFoundError: No module named '_sqlite3'`	Install the package 'libsqlite3-dev' before building python.
`Python-env-health-check fails if pandas version is less than 1.4.1.` **NOTE:** `modin dataframe library is supported only from pandas version 1.4.1 and higher.`	You can switch between the options **modin[dask]** and **pandas** for the underlying dataframe library if pandas version 1.4.1 is installed.
`"Not a valid file"` error while profiling the Hive Data sources.	Copy the following required files **: kbank.keytab , krb5.conf , hive-jdbc-driver.jar** into the path **: $DS_HOME/conf folder of datastudio**

4.3 View a Dataset

To edit a Dataset, follow these steps:

1. Navigate to Dataset Summary Page.
2. Click ••• next to corresponding Dataset and select View .

4.4 Edit a Dataset

To edit a Dataset, follow these steps:

1. Navigate to Dataset Summary Page.
2. Click ••• next to corresponding Dataset and select Edit .

   You can edit the Dataset fields except Code and Dataset Name.

4.5 Delete a Dataset

To delete a Dataset, follow these steps:

1. Navigate to Dataset Summary Page.
2. Click ••• next to corresponding Dataset and select Delete .

4.6 Create a Profile

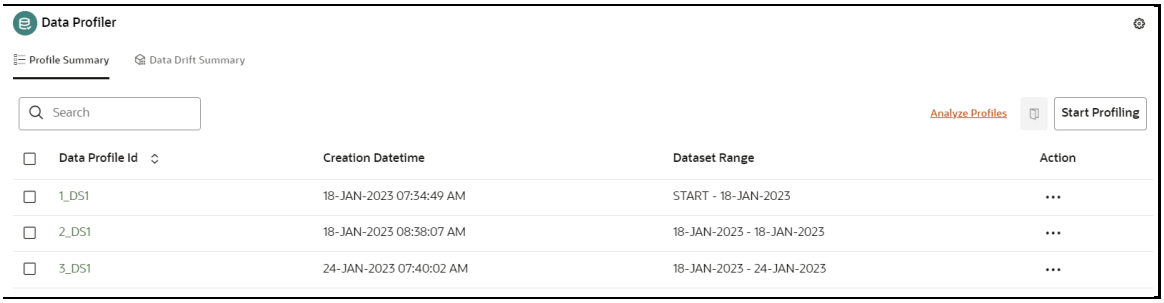
Prerequisite:

You must create a Dataset with a datasource having columns with **DATA_TYPE** as **DATE**.

To create a Profile, follow these steps:

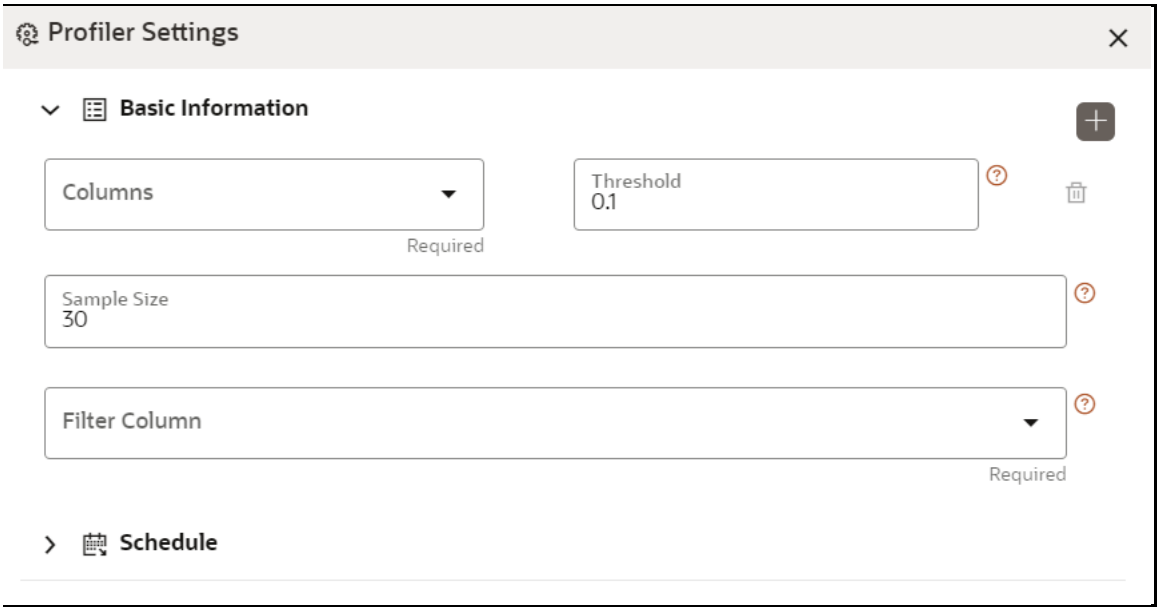
1. Navigate to Dataset Summary Page.
2. Click ••• next to corresponding Dataset and select Profile.

   This window displays the Dataset profiles in a table.



3. Click Start Profiling or click Settings icon.

   The Profiler Settings page is displayed.



4. Under Basic Information section, enter the following details:
   a. Select the required columns from the drop-down. All the numeric options are displayed under the Columns drop down. You can select multiple columns by clicking on + icon.

b. Enter the threshold for the columns. Threshold defines the Drift detection threshold for each column and only numerical columns can be selected for profiling.

c. Enter the sample size. Sample size is percentage of total rows in dataset included for creation of profile. For example, if you have 100 records in the table and want to randomly pick 30 records, enter 30 in the Sample Size field. Please only enter numerical between 0 to 99.99.

d. Select the required filter column from the drop-down. Filter column contains date column of the dataset to filter.

5. Under Schedule section, enter the following details:

a. Enter a Schedule Name.

b. Select the Schedule Type as required:

- Daily : Select to run the schedule everyday
- Weekly: Select to run the schedule once in a week or the selected days in a week
- Monthly : Select to run the schedule once in a month or the selected days in a month.

c. Click Save.

The Data Profiling has been started and Stop Profiling option is displayed in the Profile Summary screen.

For more details on disabling dataset profile, see Stop Profiling section.

4.6.1 Schedule Daily

To schedule the profiler to run daily, perform the following steps:

1. In the Profiler Settings screen, select the Schedule Type as Daily.
2. Select the start date on which you want to run the profiler.
3. Select the end date on which you want to stop your schedule.
4. Enter the time at which you want to run the profiler.
5. Click Save.

4.6.2 Schedule Weekly

To schedule the profiler to run weekly, perform the following steps:

1. In the Profiler Settings screen, select the Schedule Type as Weekly.
2. Select the start date on which you want to run the profiler.
3. Select the end date on which you want to stop your schedule.
4. Enter the time at which you want to run the profiler.

5. Select the day on which you want to run the profiler. You can select multiple days to run the profiler.
6. Click Save.

4.6.3 Schedule Monthly

To schedule the profiler to run monthly, perform the following steps:

1. In the Profiler Settings screen, select the Schedule Type as Daily.
2. Select the start date on which you want to run the profiler.
3. Select the end date on which you want to stop your schedule.
4. Enter the time at which you want to run the profiler.
5. Select the month on which you want to run the profiler. You can select multiple months to run the profiler.
6. Select the date on which you want to run the profile of the selected month to run.
7. Click Save.

4.6.4 Stop Profiling

To stop the profiling, perform the following steps:

1. In the Profiler Summary screen, select the data profiles that you want to stop profiling. You can select multiple data profiles.
2. Click Stop Profiling.

   A confirmation pop-up window is displayed.
3. Click Yes to disable the dataset profile.

   The dataset profiling has been disabled and Start Profiling option is displayed. For more information on profile, see Create a Profile section.

4.6.5 Analyze Profiles

To analyze multiple data profiles, perform the following steps:

1. In the Profiler Summary screen, select the data profiles that you want to analyze profiles. You can select multiple data profiles.
2. Click Analyze Profiles.

   A window is displayed with multiple metrics of the selected profiles. Click on each metric to view the data in graphical format.

4.6.6 Compare Profiles

To compare the data between two data profiles, perform the following steps:

1. In the Profiler Summary screen, select two data profiles that you want to compare.

2. Click Compare .

   The Profile Comparison is displayed.

3. In the Dataset Feature drop-down, select the feature that you want to compare.

   The properties of both the profiles is compared and displayed in a tabular format.

4.6.7 Calculate Drift

A Data Drift Summary report summarizes the changes in a dataset over time. It typically includes information for any changes in the data distribution of the selected features over the selected baseline and target date ranges.

The report is used to help identify potential issues with data quality which helps the users to take necessary actions further.

To calculate the data drift, perform the following steps:

1. In the Data Drift Summary screen, click Calculate Drift.

   The Data drift Analysis window is displayed.

2. Select the data range (From and To) dates for the baseline.
3. Select the data range (From and To) dates for the target.
4. Select the drift method from the drop-down.

   The available drift methods are:

   - Kolmogorov–Smirnov (K-S) test
     - Only for numerical features
     - Output: p_value, drift detected when p_value < threshold.
   - Kullback-Leibler divergence
     - For numerical and categorical features
     - Output: divergence, drift detected when divergence >= threshold.
   - Wasserstein distance (normed)
     - Only for numerical features
     - Output: distance, drift detected when distance >= threshold.
   - Population Stability Index (PSI)

- For numerical and categorical features
- Output: psi_value, drift detected when psi_value >= threshold.

- Jensen-Shannon distance
  - For numerical and categorical features
  - Output: distance, drift detected when distance >= threshold.

5. Click **Calculate**.

   The drift analysis report is displayed.

6. Click **Save**.

   The reports are displayed under Data Drift Summary screen.

4.6.7.1 Viewing the Data Drift Summary Report

To view the Data Drift Summary Report, perform the following steps:

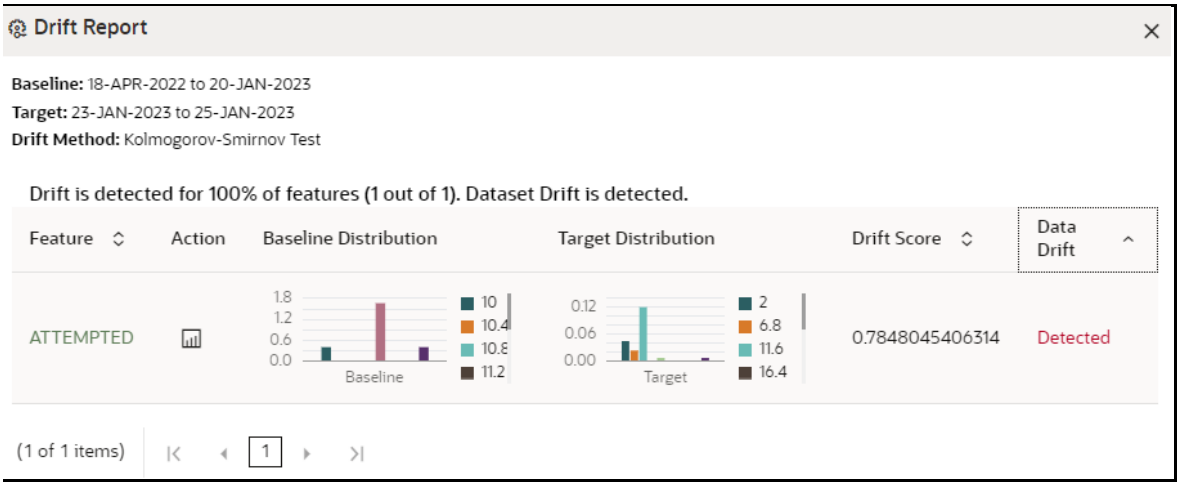
1. Navigate to **Data Drift Summary page.**

   The data drift summary reports are displayed in a table.

2. Click **Action** ••• next to corresponding drift report Id and select **View** .

   The drift report is displayed in a graphical format.

Figure 24: Drift Report



4.6.7.2 Deleting the Data Drift Summary Report

To delete the Data Drift Summary Report, perform the following steps:

1. Navigate to **Data Drift Summary page.**

   The data drift summary reports are displayed in a table.

2. Click **Action** next to corresponding drift report Id and select **Delete**  .

   OR

   If you wish to delete multiple data drift reports, select reports and click  .

   A confirmation pop-up window with report details is displayed.

3. Click **Delete**.

   The Data Drift Report is deleted.

4.7 Python functions for accessing Dataset from Model Pipelines/Notebooks

4.7.1 Listing tags of Datasets Cached from UI

To get a list of all snapshots/tags of a dataset cached from UI, use the API below:

API	Notebook Script	Notes/Input
List Tags of a Dataset	from mmg.datasets import list_tags list_tags(dscode)	dscode = dataset code - string tag = tag with which the dataset was cached - string

4.7.2 Deleting Cached Dataset

A cached dataset can be deleted by using the API below where tag refers to a particular snapshot/timestamp of the dataframe

API	Notebook Script	Notes/Input
Delete Dataset	from mmg.datasets import delete_df delete_df(dscode, tag)	dscode = dataset code - string tag = tag with which the dataset was cached - string

4.7.3 Listing all Datasets with Metadata saved from UI

API	Notebook Script	Notes/Input
List Datasets	from mmg.datasets import list_datasets list_datasets()	

4.7.4 Fetching Dataset in Notebook

Dataset whose metadata has been saved from UI can be fetched in two ways:

1. Dataset that has already been cached from UI can be fetched using first API below by providing the dataset code and tag.(DataFrame will not be recalculated. It will be read from cache.)
2. A new data frame for a dataset can be calculated/fetched using data from the present time with the metadata saved for that dataset as given in second API.

API	Notebook Script	Notes/Input
Fetch Dataset Cached from UI	from mmg.datasets import fetch_ds df = fetch_ds(dscode,tag)	dscode = dataset code - string tag - string
Fetch New Snapshot of dataset	from mmg.datasets import fetch_ds df = fetch_ds(dscode)	dscode = dataset code - string

4.7.5 Cache User's Dataframe from Notebook

A user-made data frame can also be cached using the below API. It will be stored in the datastudio server in location : $DS_HOME/work/ftpshare/mmg/workspace_name/cached/tag. The dataset will be saved as a parquet file with name cached_tag.parquet.

Note: This dataframe is not related to dataset created from UI. This is independent of dataset metadata.

API	Notebook Script	Notes/Input
Caching Data Frame	from mmg.datasets import cache_df path=cache_df(df, tag) path	df = dataframe to be cached - pandas dataframe tag - string

4.7.6 Fetching Data Frame Cached from Notebook

The data frame cached from notebook can be fetched using the below API.

API	Notebook Script	Notes/Input
Fetch dataset cached manually	from mmg.datasets import fetch_ds df = fetch_ds("cached",tag)	tag - string

4.7.7 List Tags of all Data frames Cached from Notebook

API	Notebook Script	Notes/Input
List Tags of Manually Cached Datasets	from mmg.datasets import list_tags list_tags()	

4.7.8 Sample Custom Scripts

Below are few sample scripts which the users can refer to create transformations.

Function	Script	Comments
Directly pass input data frame to output	df_out=df_prev	
Drop first row of data frame	df_out=df_prev.drop(0,axis=0) if not df_prev.empty else df_prev	
Drop column from data frame	df_out= df_prev.drop('colname',axis=1)	FEATURE TRANSFORMATION

4.7.9 Transformations

These are the various transformations which can be done from the UI.

No.	Transformation	Function
1.	Add New Feature	A new feature can be added to the dataset which could be derived from the existing features using Script. Physical Feature Name and Feature Name are the names of the new feature. Script can be used to create a pandas Series for the new feature
2.	Encode Categorical Features	This function performs One Hot Encoding on a categorical feature and replaces it with multiple numerical features in the dataset.
3.	Encode Datetime Features	This function encodes a datetime feature and replaces it with multiple numerical features having the following information derived from the datetime feature - year, month, week, day, hour, minute, dayofweek
4.	Encode Cyclical Features	This function encodes a cyclical feature having hour, minute data, and so on and returns two features carrying the sine and cosine transformation of the cyclical data. 'fmax ' denotes the maximum possible value of the cyclical feature data.
5.	Impute Missing Data	This function imputes missing data within a feature. For numerical features, there are 4 methods for imputing missing data. simple - imputes with mean, median, most_frequent values based on chosen arg value using the SimpleImputer in sklearn. const - fills the missing values with the value given in the arg knn - imputes using the KNNImputer in sklearn with k value given in arg mice - imputes using the IterativeImputer in sklearn For non-numerical data, missing values can be imputed using the 'const' method by replacing all missing values with the value given in arg
6.	Feature Scaling	This function is used to scale multiple selected numerical features using the StandardScaler in sklearn
7.	Dimensionality Reduction	This function performs PCA on selected numerical features to reduce the dimensionality using sklearn.decomposition.PCA module. The number of output features can be specified using dim field. The names of the output features' names can be specified in the fields 'Physical Feature Name' and 'Feature Name'
8.	Outlier removal	This function is used to remove outliers present in a feature based on the specified zscore value. Non-numerical features are label encoded before removing the outliers.
9.	Duplicates Removal - Data Frame	This function removes all duplicate rows in the dataframe.

No.	Transformation	Function
10.	Duplicates Removal - Feature	This function removes all duplicate rows within a specified subset of features and consequently removes those rows from the data frame
11.	Filter Features	This function is used to filter the data frame based on conditions specified on features Operations allowed : >,>=,=,!=,<,<=,isin When the chosen operation is 'isin', the input to 'Filter Value' is a list of values that should be present in the output data frame

5 Using Model Pipeline

Model Pipeline allows you to create and publish models based on the workspaces created from datasources. The published models are then deployed in production to be consumed by users. Modelers create models by using the Notebooks in the Data Studio. The Notebooks are used to create training models and the iterations of comparison between various models lead to the elimination of undesired models and filters a few robust ones that can be considered for deployment in production. Modelers then use their better judgement to consolidate their choice and fix on one model - the champion model. The champion model is also called the scoring model or the actual model in this document.

5.1 Prerequisites

The prerequisites for model pipeline are as follows:

- To create a model, your user profile must be mapped to the Modeler Group. For more information, see the Mapping User Groups section.
- To create a model, a workspace must be deployed. For more information, see the Create a Workspace section.
- To approve and deploy a model, your user profile must be mapped to the Modeling Administrator Group. For more information, see the Mapping User Groups section.

5.2 Access the Model Pipeline

The Model Pipeline window allows you to create and publish models.

To access the Model Pipeline window, follow these steps:

1. Navigate to **Workspace Summary** page.

   The page displays workspace records in a table.

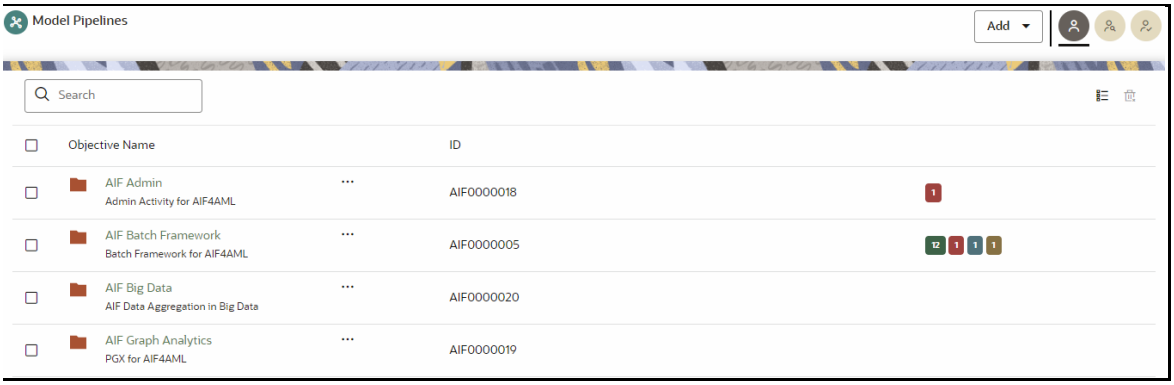
2. Click  next to corresponding Workspace to Launch Workspace.

   The MMG **Dashboard** window is displayed with application configuration and model creation menu.

3. In the LHS menu, click **Model Pipelines**  to display the **Model Pipelines** page.

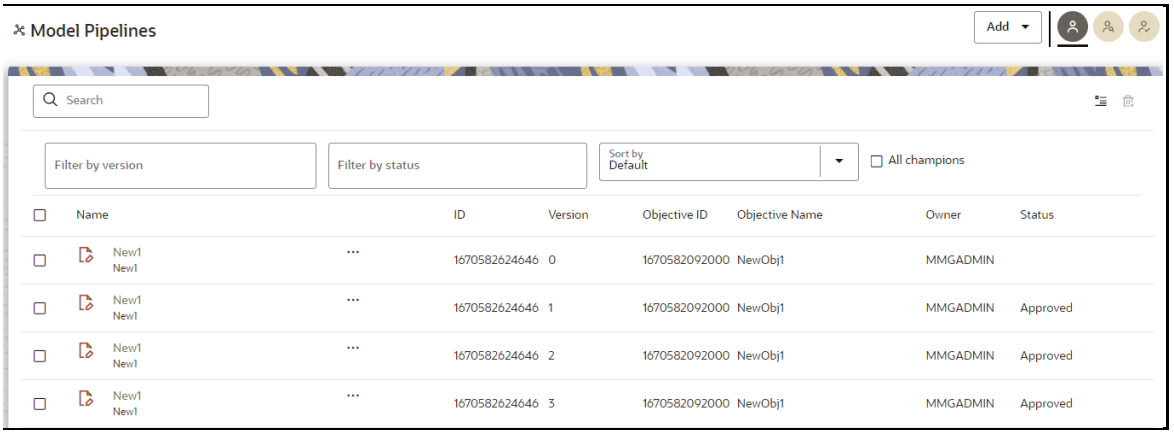
   The window displays objectives that contain drafts and models. When you hover on the count that are next to the ID column, it displays the count of sub objectives, Drafts, Models, and Champion if available.

Figure 25: Model Pipelines Page



You can switch the Model Summary page view from Flat to Hierarchical and vice-versa. Flat list is the default view in Production workspace while Hierarchical list is the default in Sandboxes.

- **Hierarchical option**: The Model Management window shows the hierarchical list of the Objectives using the Hierarchical option. In the Hierarchy view, you can see the following details of the Objectives such as Objective Name, and ID.
- **Flat option**: The Model Pipeline window shows the flat list of all the models (published and drafts) using the Flat option. Flat list is not objective-specific. It shows the models across all the objectives. You can search the models using the Filter by Version, Filter by Status, and All champions. You can also sort the drafts and models by Default or Latest first options.
  - In the Flat list of models, you can see the following details of the models such as Name, ID, Version, Objective ID, Objective Name, Owner, and Status.



The following table provides descriptions for the fields and icons on the **Model Pipeline** page.

Field or Icon	Description
Model Pipeline Page Header	The header that follows the global header in the application.
Breadcrumbs	Indicates the position of the current page in the Model Management hierarchy. Use breadcrumbs locator links to navigate back to higher levels in the hierarchy after you have drilled down through levels of functions.  Click  to navigate to Model Summary page. Click  or  to navigate back to Workspace Summary page.
Create Objectives and Models	Select Add to display a list with the following options: • Draft • Objective • Seeded Models To create Models, select Draft. To create Objectives, select Objective. See the Create Objective (Folders) and Create Draft Models sections for more information. However, if you want to know about the cycle of model creation, see the Create, Review, Approve, and Deploy a Model Section.
Requester	Displays that the logged-in user has the Requester privileges when the status is green.  You can create a model. However, to approve and publish, the model must be reviewed by a user with reviewer privileges and approved by a user with approver privileges.
Reviewer	Displays that the logged-in user has the Reviewer privileges when the status is green.  You can review models. However, to approve and publish, the model must be approved by a user with approver privileges.
Approver	Displays that the logged-in user has the Approver privileges when the status is green.  You can approve models that are created and reviewed.
Model Pipeline Page Table	The table displays the objective and model records on the page.

Field or Icon	Description
Objective Name	Displays the name and description of the Objective 
ID	Displays the ID of the objective.
Owner	Displays the owner who created the Drafts. This information does not display for an Objective.
Tags	Displays the tags associated with the Models or Drafts.  This information does not display for an Objective.
Delete Objective	Select  to delete the model from the **Confirmation** dialog box. Click **Delete** to process or click **Cancel** to cancel. This information does not display for an Objective.
Edit Objective	Select  to edit the models in Data Studio. See the Edit Models section for more information. This information does not display for an Objective.

5.3 Create, Review, Approve, and Deploy a Model

Model creation and deployment undergoes a workflow of Model Governance where the following types of users in the system have privileges that restrict the activities, they can do in the model creation and deployment workflow.

Field or Icon	Description
Requester 	You can create a model. However, to approve and publish, the model must be reviewed by a user with reviewer privileges and approved by a user with approver privileges. **NOTE:** User Groups must be mapped to the MDLDEPLOY role to access Requester functions.
Reviewer 	You can review models. However, to approve and publish, the model must be approved by a user with approver privileges. **NOTE:** User Groups must be mapped to the MDLREVIEW role to access Reviewer functions.
Approver 	You can approve models that are created and reviewed. You can then promote to production and make the model the champion in the production. **NOTE:** User Groups must be mapped to the MDLAUTH role to access Approver functions.

The following sections in this topic provide details for the cycle of creation of a model, review, approval, and deployment:

- Create Objective (Folders)
- Create Draft Models
- Publish Models (Scoring)
- View Model Details
- Compare Models
- Understand Model Governance
- Request Model Acceptance
- Review Models and Move to Approve or Reject
- Approve Models and Promote to Production
- Deploy Models in Production and Make it a Global Champion

5.3.1 Create Objective (Folders)

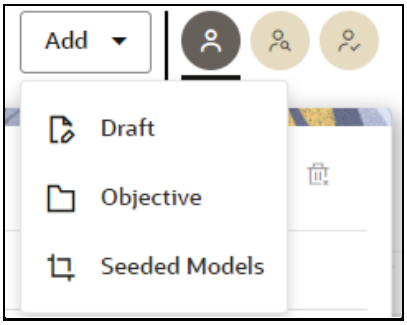
Create folders called Objectives within which you can create Models.

To create an Objective, follow these steps:

1. Click Launch Workspace  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines**  to display the **Model Pipelines** window.

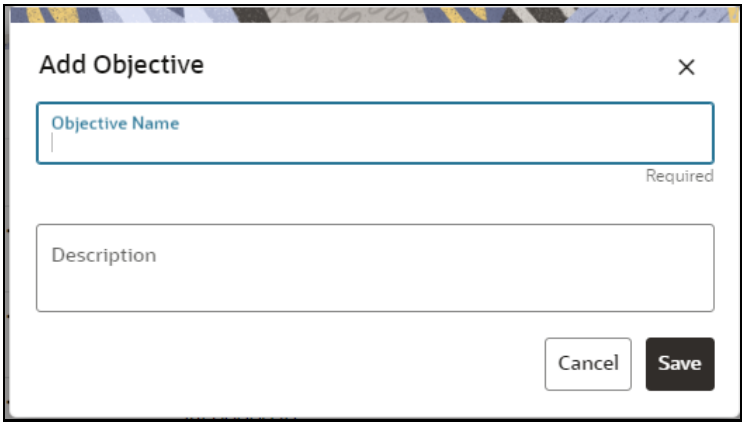
   The window displays folders that contain models and model records in a table.
3. Click **Add** and select **Objective** from the list to display the **Objective Details** dialog box.

Figure 26: Select Objective from Add



4. Enter details in Objective **Name** and **Description** fields in the Add **Objective** dialog box.

Figure 27: Objective Details Dialog box



5. Click **Save**.

5.3.2 Create Draft Models

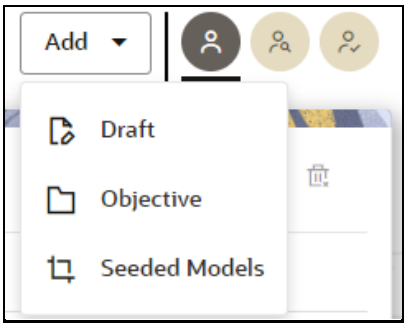
Create Models that are classified as draft models. These models will be reviewed before being sent for Scoring.

To create a draft Model, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines**  to display the **Model Pipelines** window.

   The window displays folders that contain models and model records in a table.
3. Click **Add** and select **Draft** from the list to display the **Add Draft** dialog box.

Figure 28: Create Model



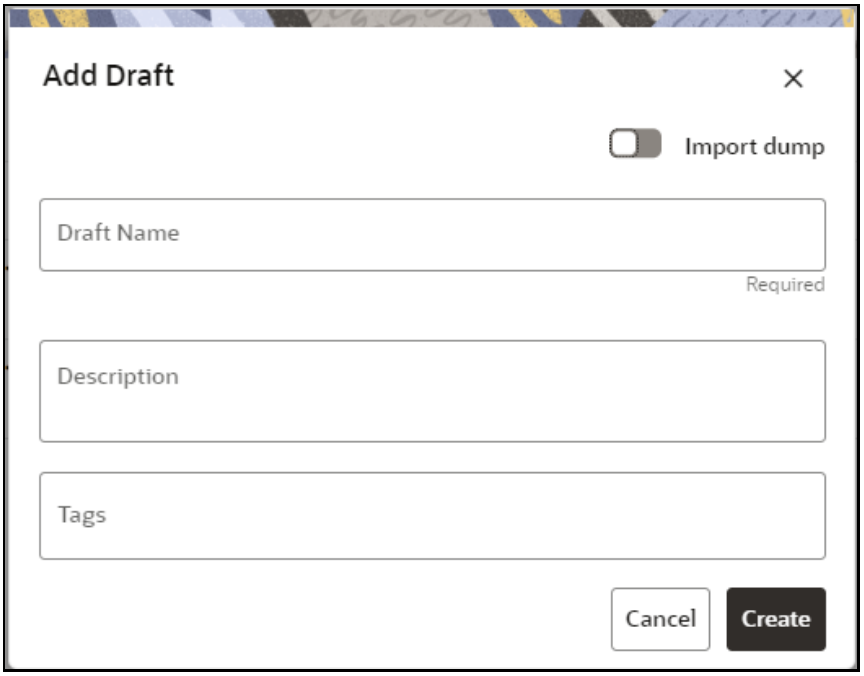
4. **Create New Model** is the default setting in the **Model Details** dialog box. Drag the toggle button to select **Import Dump**. Use Create New Model to start from a blank Notebook in Compliance Studio. Import Dump lets you drag and drop an existing file with model data and modify it. To import a model data dump from another

model, see the Import a Workspace Model Data into a New Model section. You can also create a draft under Objective (Folder) also. Click an Objective to open it.

To create a new model, follow these steps:

a. Enter details for Draft **Name** and **Description**.

Figure 29: Model Details - Create New Model



b. Enter a tag in the **Tags** field.

c. Click Create.

5.3.3 Create Seeded Models

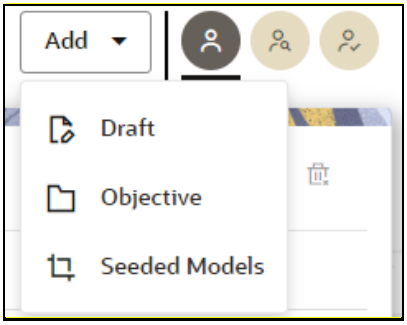
You can seed the models from the external sources which can be imported in the MMG application.

To import the models, follow these steps:

1. Click **Launch Workspace** next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines** to display the **Model Pipelines** window.

   The window displays folders that contain models and model records in a table.
3. Click **Add** and select **Seeded Models** from the list to display the **Add Draft** dialog box.

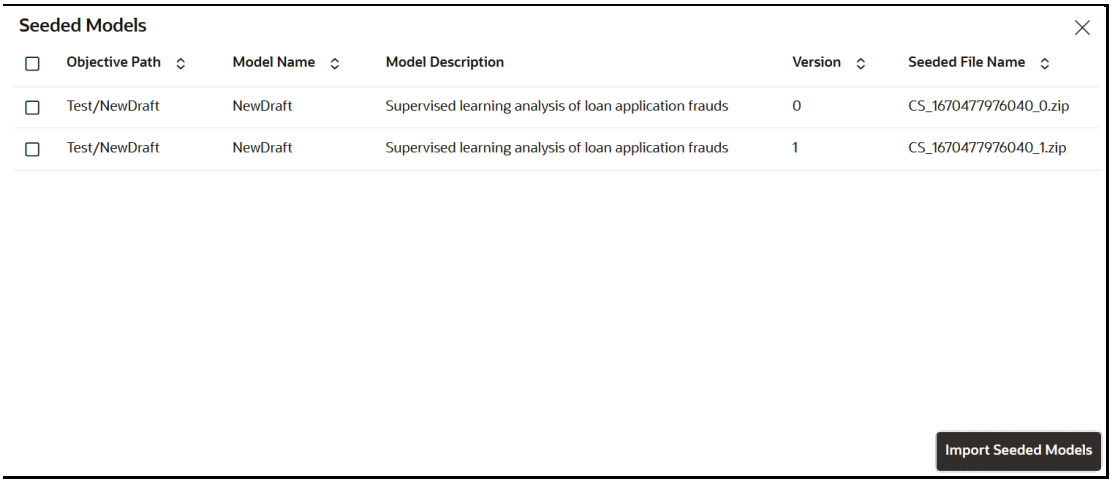
Figure 30: Add Seeded Models



4. You must add the models in the following installed path location:
   `/scratch/ofsaaweb/ftpshare/mmg/seeded/models`

   The added models is displayed in the Seeded Models page.

Figure 31: Seeded Models



5. Select the models which you want to import and click **Import Seeded Models**.

   The selected models are imported and displayed in the **Model Pipelines** page.

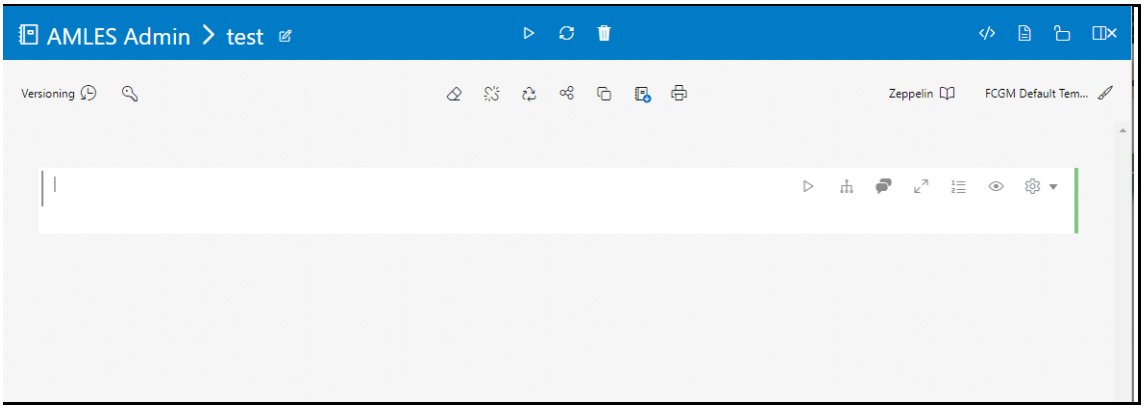
5.3.3.1 Create Paragraphs in Model Studio Notebooks

After creating the Models in the Workspace, create Paragraphs in the Model Studio window. To create Paragraphs, you must have a working knowledge of scripting and Python.

The following types of paragraph creation are supported for model creation:

- Add empty Paragraph:  To create Model building scripts in Python.

Figure 32: Create Draft Models in Model Studio Notebooks



To create Paragraphs in the Model Studio Notebooks, follow these steps:

1. Open Model Studio in OFS MMG. Follow the instructions in the Create Draft Models (Draft) section to open it.

2. Click  in Model Studio to add a Paragraph in the Notebook and add the scripting instruction to fetch connection objects in the following format. This creates a cx_oracle based connection object to the datadom of the infodom being passed:

   ```
   conn = mmg.getConnection(<infodom name>)
   ```

   > **NOTE** The fetch connection objects function works only with %Python interpreters and requires an Oracle client installed on the server that runs Model Studio.

   Click  to run the script and display the Notebook ID and the Model Objective ID.

3. Click  in Model Studio to add a Paragraph in the Notebook and add in the following format to fetch current Notebook and Model Objective details:

   - Notebook Id: **currentNotebookId**
   - Model Objective: **objectiveId**

   Click  to run the script and display the Notebook ID and the Model Objective ID.

4. Click  in Model Studio to add a Paragraph in the Notebook and fetch the runtime parameters supported in Model Studio runtime as shown in the following example. This is available for all the interpreters.

For example, enter as follows:

```
%python
print('threshold value is : ${[threshold]}')
```

Click ▷ to run the script and display.

After the Draft Models are created, publish the Notebooks to create Scoring Models. See the Publish Models (Scoring) section for more information.

5.3.3.2 Create Paragraphs using Pipeline Designer

After creating the Models in the Workspace, create Paragraphs using the Pipeline Designer window.

Pipeline Designer enables you to design the paragraph using widgets (graphical representation) instead of using python codes. In addition, if you add new paragraphs in Data Studio, the added paragraphs are displayed in the widget format on Pipeline. Similarly, if you create a Notebook using Pipeline Designer, it can be opened for editing in Data Studio using Studio Notebook option.

> **NOTE** When you open the notebooks from MMG UI, the attach workspace call will be made from mmg service and proper workspace will get attached.
>
> If the Studio is opened outside of mmg, then the attach_workspace command has to be used.

This helps the Financial Institutes and Banks in following ways:

- Visualization of the data (for example, based on data tasks)
- View the dependency
- Modify the flow of execution or execution order
- Easy for Auditing purpose

You can execute the flow based on requirement. For example, if you have created one flow and want to execute a flow of training paragraphs out of that and other flow as experimental way, then you can modify using the Training and Experimentation link types. One flow can be break into 2-3 flows for execution purpose.

When a draft is edited using the Pipeline Designer/Data Studio, and published, then a new version of published model is displayed in Model Summary page.

NOTE

When you add a new paragraph from studio and opened the same in pipeline designer, it gets linked to multiple paragraphs. For example, you have paragraphs P1, P2 and P3 in the same order in a Notebook. It shows in Pipeline Designer canvas as P1 > P3 > P2.

If you add two new paragraphs P1a and P1b in the Notebook after the P1 and open the canvas, this gets reflected as the following:

a. 1. P1 -> P1a -> P1b -> P2

b. 2. P1 -> P3 -> P2

After opening a Model in the Pipeline Designer, following options are displayed:

- Model Report
- Download
- Delete
- Clone Model

After opening a Draft in the Pipeline Designer, following options are displayed:

- Model Report
- Download
- Delete
- Publish

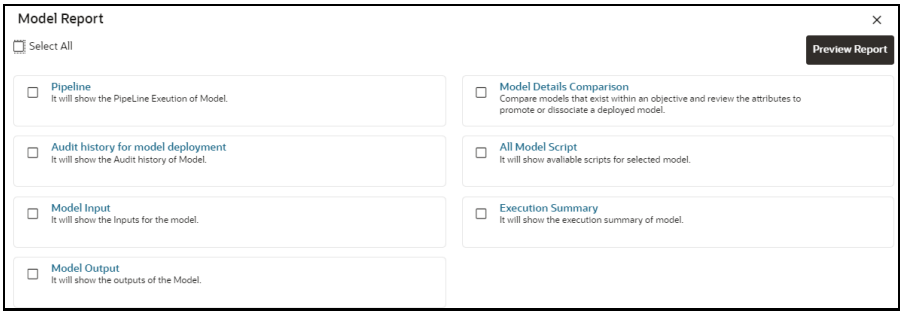
5.3.3.2.1 Model Report

This option allows you to view the Model report and download the same in PDF format.

To view and download Report, follow these steps:

1. Open a Model in Pipeline Designer.
2. Click **Model Report**.

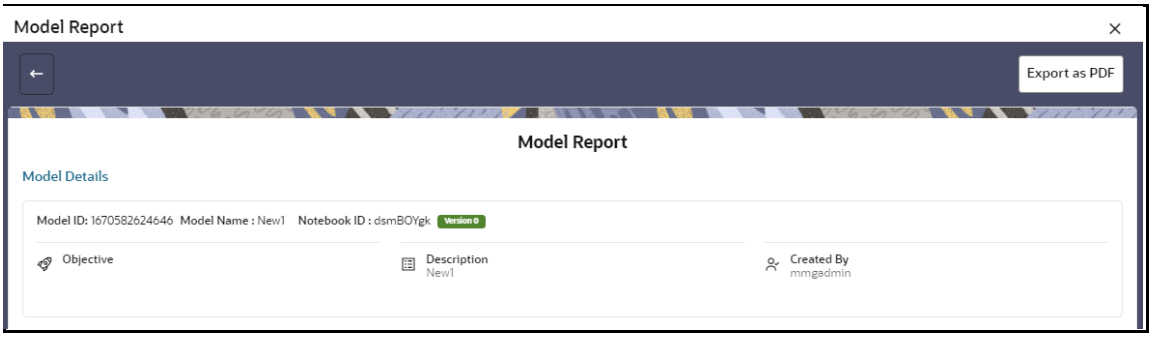
   The Model Report window is displayed.



3. Select the Parameters to generate the report.

4. Click **Preview Report**.

   The report is displayed based on selected parameters.



5. Click **Export as PDF** to save the report as PDF in local system.

5.3.3.2.2 Download a Model

This option allows you to download the Model. To download a model, follow these steps:

1. Open a Model in Pipeline Designer.
2. Click **Download**.

   A zip folder is downloaded. This folder contains .cfg and .dsnb files.

5.3.3.2.3 Delete

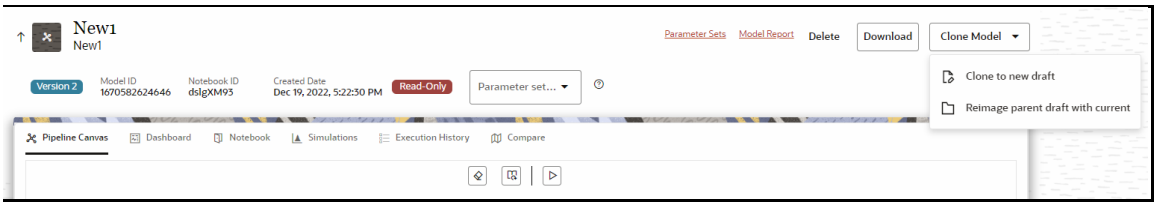
This option is used to delete the Model.

5.3.3.2.4 Cloning a Model

You can pick any published model and clone the contents to a new draft in the same objective or clone the content to the current parent draft. The cloned draft can be edited and used further. Audit Trail window also captures the clone information.

To clone the model details, follow these steps:

1. Open a Published Model in Pipeline Designer.



2. Select Clone to new Draft or Reimage parent draft with current.

5.3.3.2.5 Pipeline Designer

The following sections are available on the Pipeline Designer window:

- Pipeline Canvas
- Execution History
- Notebook
- Dashboard
- Compare
- Simulations

NOTE:

a. Models in Production workspace have the 'Dashboard' as the default tab.

b. Drafts or Models in the Sandbox workspace have the 'Pipeline Canvas' as the default tab.

5.3.3.2.6 Pipeline Canvas

You can perform following functions on Pipeline Canvas:

- Creating a Pipeline
- Creating Script Template
- Viewing a Pipeline
- Using Link Connector Nodes
- Execution of Pipeline

Creating a Pipeline

To create a paragraph using pipeline, follow these steps:

1. Navigate to the **Pipeline Designer** page. For more information, see the Creating Draft Models section.
2. Drag and drop the Paragraph widget from the widgets pane in the upper-left corner of the designer pane.

   The 'START' widget is displayed by default in the canvas screen. You can edit this Node except script. Whenever a new draft is created (not by importing dump files), the default paragraph created is converted into a start widget. The visibility of code/result/title in notebook of this node will be kept to invisible.

   Whenever the notebook is opened, init script execution including workspace attach will happen in this start node.

   Title of the node is called 'Start widget'. Publish /download and import /promote to production of model with start widget will keep this widget. Publish from canvas will keep the start widget in the published model. When you publish model from model

summary, you can explicitly select the start widget paragraph from the list of paragraphs. After this only, start widget will appear in the published model.

3. Hover the mouse over the Paragraph widget and click  to add Basic Details.

Figure 33: Pipeline Designer

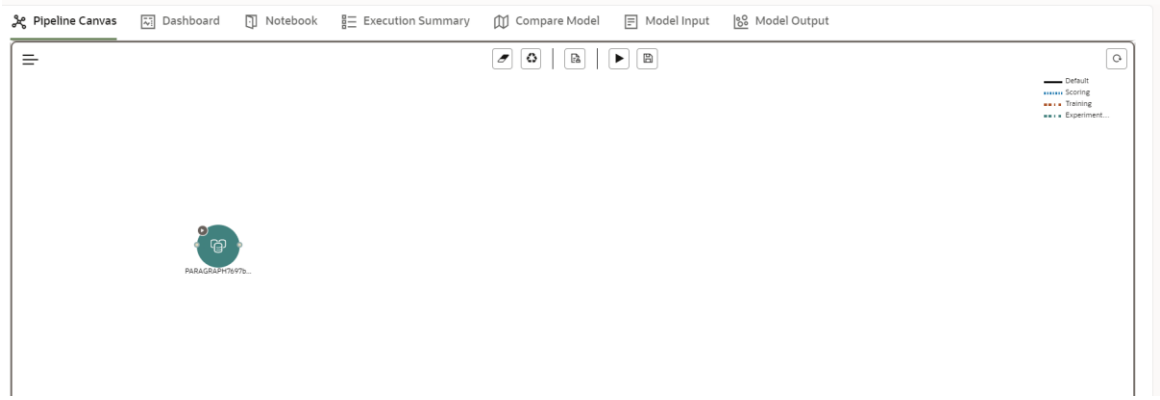
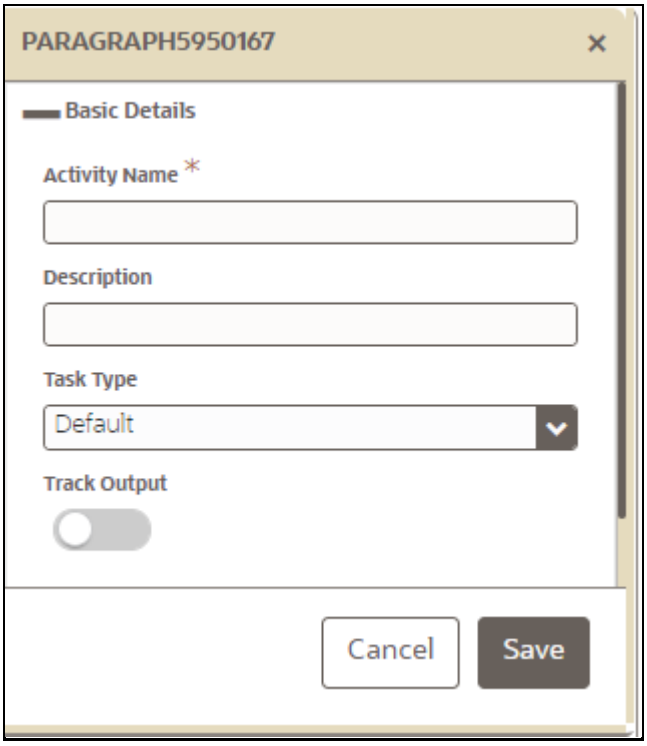


Figure 34: Basic Details for Paragraph



4. Provide details as described in the following table:

Field	Description
Activity Name	Enter the Activity Name
Description	Enter the description of Activity
Task Type	Select the task type. For example, Model Training, Data Analysis and so on. You can also search the task type.
Track Output	If this option is selected for any paragraph, then during the model comparison the output details are displayed. Keep the Track Output to ON in case you want to execute the paragraph and view the result from the Dashboard tab.
Script	Shows the script.

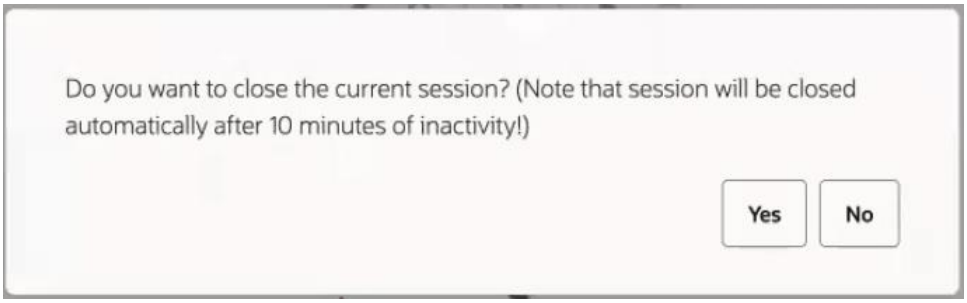
You can perform the following functions on Pipeline Canvas window:

- Clear Execution Results: Clears the execution details.
- Re-initialize session: Allows you open a new session.
- Publish: Allows publishing the Notebook.
- Save Now: Allows saving the Notebook.

- Delete: Click to Delete icon to delete the current working version. If this is first draft of Model, it will delete all the dependent published version in the Sandbox. If the Model is not first version, then it will delete only the current working version.
- Download: Use Download to download the current working version in opened in canvas
    - Execute: Allows to execute the Notebook. For more information, see Executing a Pipeline

Whenever user executes a batch, a user session is created. This execution time can be less or more for any execution. Sometime, user doesn't want to wait for execution to complete and navigate away from the Pipeline Canvas page.

For example, if user doesn't want to execute all the paragraphs and want to execute only Paragraph 1 and Paragraph 2. Paragraph 1 takes 15 mins time for execution and Paragraph 2 takes 10 mins for execution. Paragraph 2 execution also wants to use the execution of Paragraph 1. In this case, user can navigate away from the page. A confirmation message is displayed to close the current session. Here, this session time is configurable.



- If user clicks Yes, then execution thread will be closed after for this given session time.
- If user clicks No, then execution thread will be valid for this given kill time and run in the background.

User can configure these values in Application.xml file. For more information, see the MMG Installation Guide.

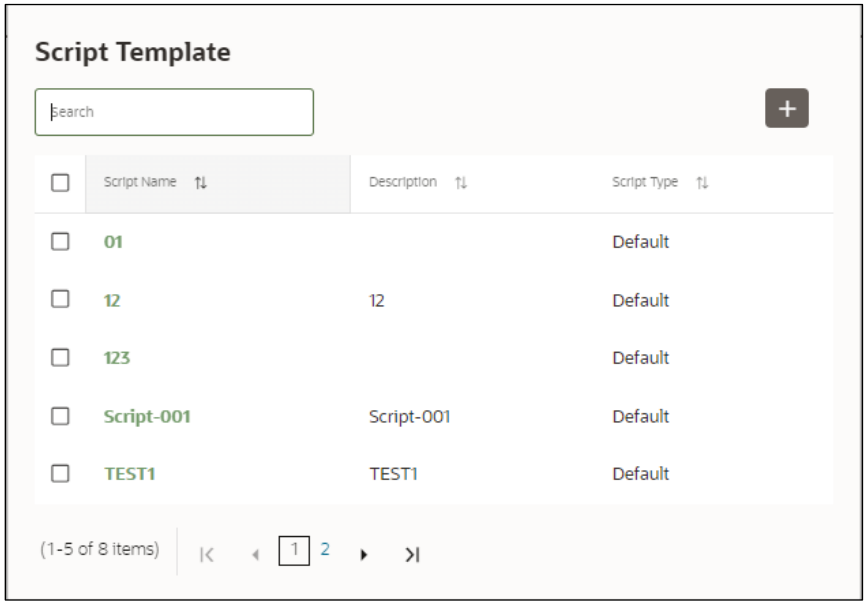
Viewing a Pipeline

The Pipeline canvas window allows you to view the Pipeline using following options:

- Auto-align : Arrange all the widgets in vertical order. After saving, the reverting option will not work.
- Revert-align : Revert all the widgets if they are Auto-aligned.
- Hide/Show Overview : Shows or Hides the overview of widgets.
- Show Full Labels : Shows the full name of activity. By default, it shows only 14 characters of label name.

Creating Script

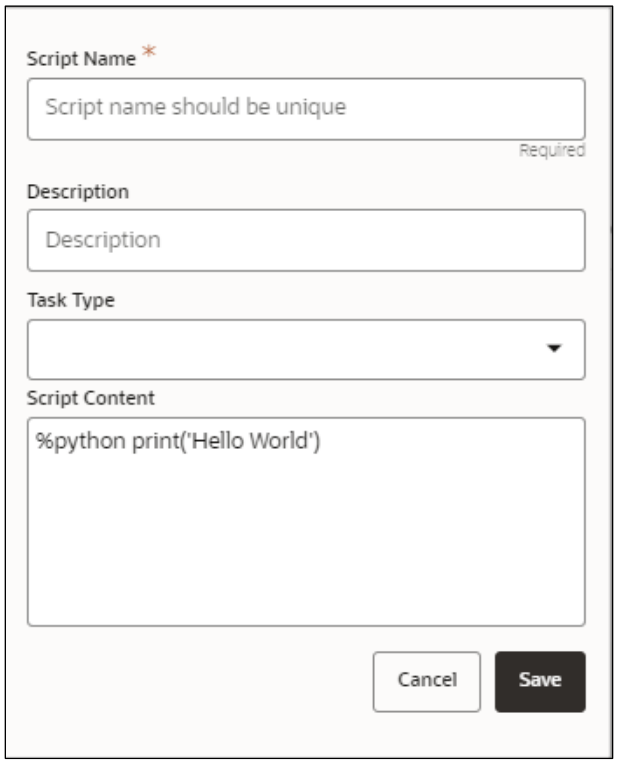
The Script Template window allows you to manage scripts. These scripts can be called for paragraph.



To add a script, follow these steps:

1. Click Tasks to navigate to Script Template.
2. Click Add in Script Template window.
3. Enter the following details:

Field	Description
Script Name	Enter the Script Name
Description	Enter the description of Script
Task Type	Select the task type. For example, Model Training, Data Analysis and so on. You can also search the task type.
Script Content	Enter Python script



You can select Parameter set and link type for execution in the canvas dashboard tab. This parameter set is available from already saved list. When you select this and there are no common keys, it is displayed along with the original parameters. If there are any common keys, it will be replaced with the chosen parameter set keys.

A reset button returns to original parameters.

Using Link Connector Nodes

You can use a dummy node that helps to service connecting functions. If a paragraph in studio got deleted, this type of widget will be placed instead of the paragraph widget in pipeline canvas Node name for deleted node 'Connector'. This helps if the links are not broken. You can drag and drop this node but will not be associated with any paragraph. If required, you can delete it and provide name and description for it.

To use Link connector nodes, follow these steps:

1. Click Tasks icon.
2. Select Link Connector and click . This is a dummy node with no paragraph is created/associated on node save. During execution, this is used in to execute API, but will not get executed. It behaves like non-physicalized paragraph widget on execute.

   This node can be used as a dummy start node or connector node.

Executing a Notebook

The Execute icon on Pipeline Canvas allows us to execute the notebook.

The following link types are available in the Pipeline Designer:

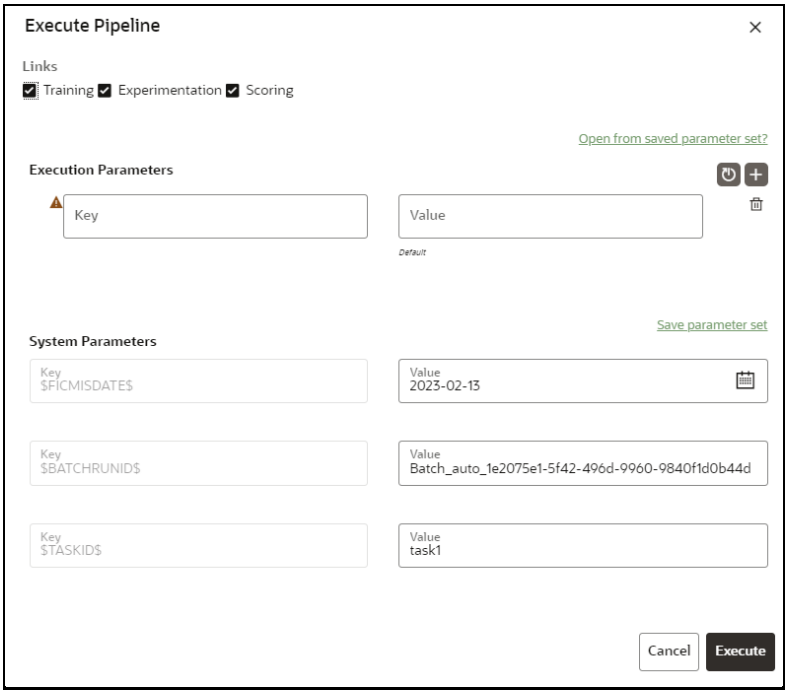
- Default
- Scoring
- Training
- Experimentation

Note: When a model gets published from Model summary page, the Link types configured in Pipeline Designer are set to Default link type.

To execute the notebook, follow these steps:

1. Click Execute to view Execute Pipeline window.

Figure 35: Execute Pipeline



2. Execution parameters are the parameters defined in the notebook required for execution. Select the flow, which you want to execute Scoring, Training, and Experimentation. It displays all the keys defined for all the paragraphs in the notebook with a placeholder for providing the values.
3. Enter the execution Key and Value.

   You can also use Runtime parameters for execution. This runtime parameter must be defined in Notebook. If this is defined, you can enter execution value during the process execution.

   For more information, see Create Paragraphs in Model Studio Notebooks section.

The System Parameters window also shows the execution ID, execution Date, and execution Batch. These are required for executing all the paragraphs along with other parameters. It also shows from where the parameter comes from as a subscript.

- **Parameter Sets:** These are the set of parameters with a specific value required for an instance of execution. It consists Key and Value. You can save the parameters set that can be used for one execution instance and reuse it for the next execution. It consists of parameters with a specific value to each parameter. Parameters containing no value will not be taken. Each set is identified with a unique code for each objective. While saving the parameter, you have to provide a code for identifying the name and description which is not mandatory.

  You can save Key Value parameter set using the Save Parameter Set option. To Save Parameter Set, enter the Threshold Value and Description in the Parameter Set window.

- **Selecting Parameter Set:** These saved Parameter Set can be selected during the execution. It will replace the values of the parameters from the chosen Parameter Set.

  To select the Parameter Set, follow these steps:

  i. Click “Open from saved Parameter Set". The Threshold Code window is displayed.

  ii. Select the Parameter Set from the available list. You can select multiple Parameter Set in the same execution instance. In that case, if there are any common keys, value will be replaced with that from the latest Parameter Set selected.

4. You can add new parameters using **Add**  .

   Note: If the parameter is not defined in the notebook, it will not be used for the execution. In case of multi select, if there are common parameters among the chosen scenarios, it will take the value based on the order of selection. that is first chosen scenario parameter will be taken.

5. But if open from saved Parameter Set again (not on single go), then already added will get replaced by the newly added (same as what existed)

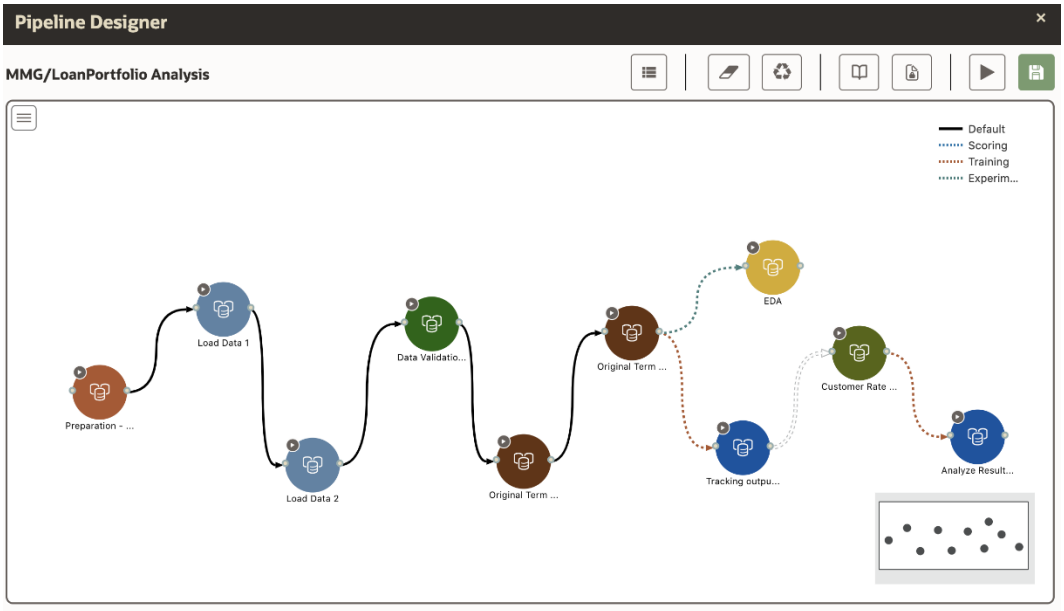
6. Execution is performed based on selected link types. It filters out all the not required/unused parameters. And all the unused parameters for the current execution are displayed with a warning  . To view the only required parameters, click **Show only required** link.

7. Click Reset  to reset the entered data.

8. Click Delete  to delete the entered Key and Value.

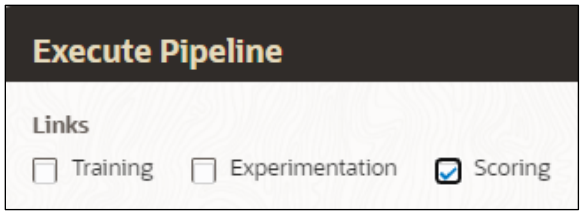
For example, refer to below Figure.

Figure 36: Example of Pipeline Designer



- Here, if you want to execute this Notebook for scoring purpose, then the flow will be executed till EDA paragraph with default paragraphs. To perform this, Click Execute Notebook and select Links as Scoring.

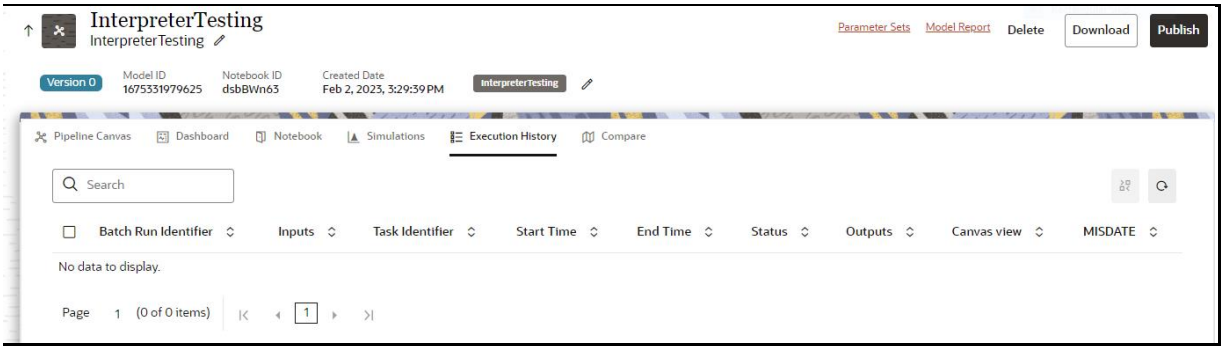
Figure 37: Execute Pipeline



- Similarly, if you want to execute this Notebook for training purpose, then the flow will be executed till Analyze Result paragraph with default paragraphs. To perform this, Click Execute Notebook and select Links as Training.

5.3.3.2.7 Execution History

This section of Pipeline Designer shows the history of the executed pipelines.



Comparing Execution

The Compare option allows you to compare the two executions.

To compare, follow these steps:

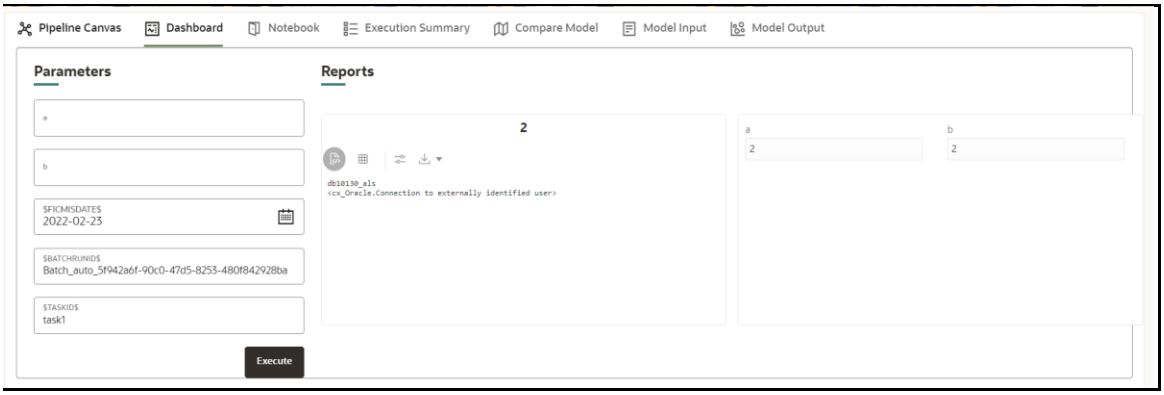
1. Navigate to Execution Summary window.
2. Select the executions using the corresponding checkboxes.
3. Click 

   The Execution Comparison window is displayed.

5.3.3.2.8 Dashboard

This section of Pipeline Designer shows the Model Dashboard to view the execution output of Model. You can also execute the Models from here.

NOTE: For Promoted models, the 'Dashboard' is the default tab.



> **NOTE** There is no Cancel button in Settings tab of reports in the Dashboard tab. You can press 'Escape' key to close the Settings.

5.3.3.2.9 Compare Models

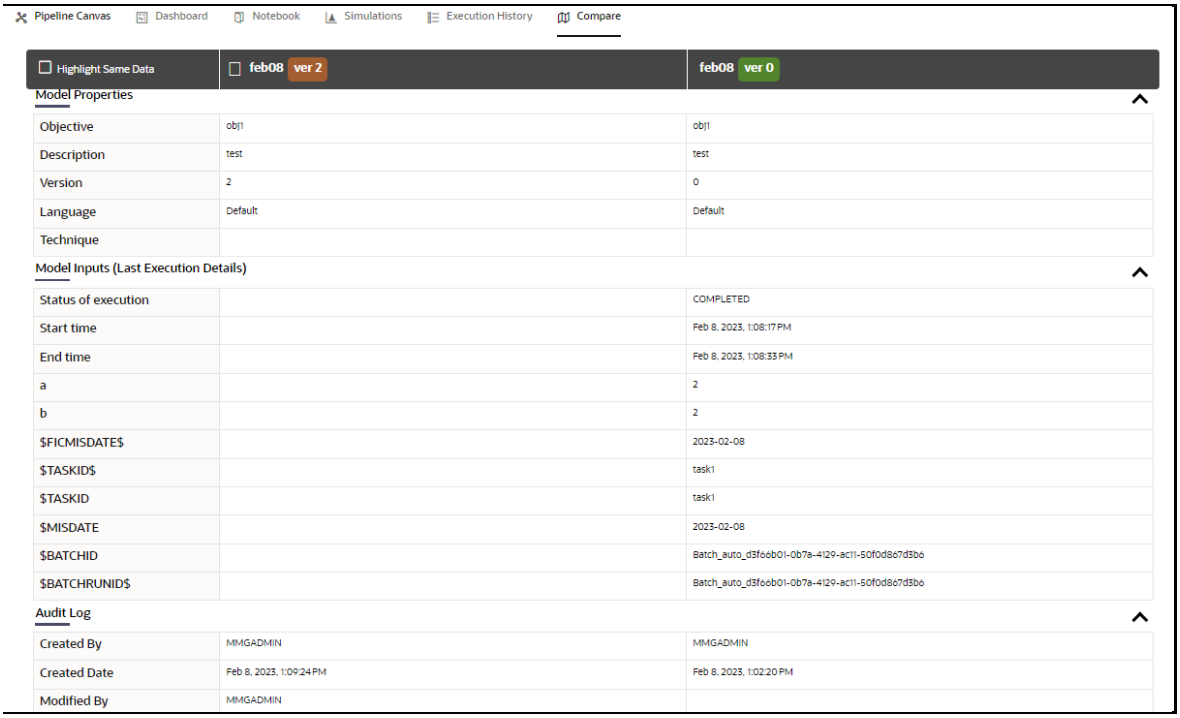
This section allows you to compare the models with Champion model.

To compare, follow these steps:

Navigate to Pipeline Designer window and click Compare Model window.

This shows the following comparison details:

- Model Properties
- Model Inputs (Last Execution Details)
- Audit Log
- Model Script
- Audit Trail



5.3.4 Publish Models (Scoring)

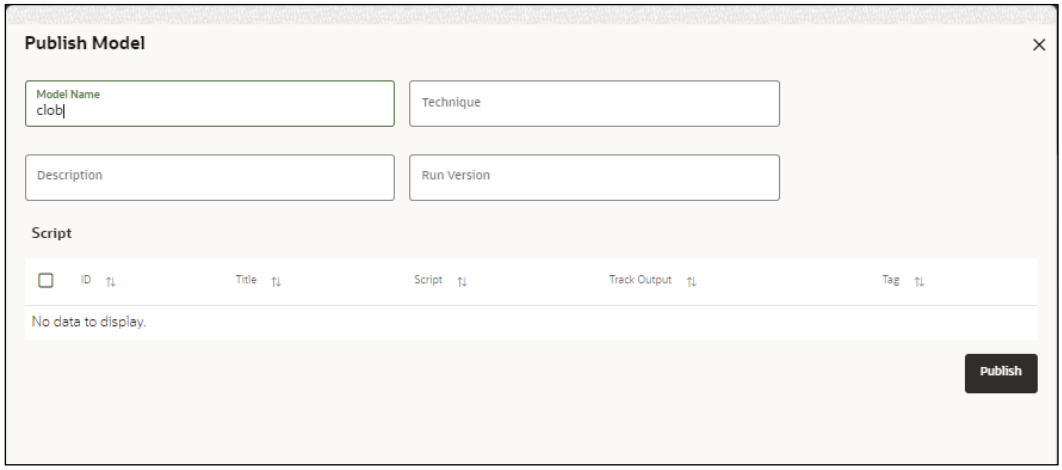
After creating the Draft Models, publish the Notebooks, which have the Model script.

To create a Scoring Model, follow these steps:

1. Create a Draft Model. See the Create Draft Models section for more information.

2. Click  next to corresponding Draft Model and select Publish Data Studio  to Open Model window.

Figure 38: Publish Model



3. Enter the details as shown in the following table.

Field or Icon	Description
Model Name	The field displays the name of the Model. Modify the name if required.
Model Description	The field displays the description for the Model. Enter or modify the description if required.
Technique	Enter the registered technique to use.
Run Version	Select a run version.
Script	The table displays the Paragraphs created in the Training Model. Select the Paragraphs that you want to use to create the Scoring Model. Track Output - Select this to track the output of the paragraph.

4. Click **Publish**.

5.3.5 View Model Details

You can view model information for deployed models, models that require approval, and so on.

To view model details, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines**  to display the **Model Pipelines** window.

The window displays folders that contain models and model records in a table.

- The  icon indicates that Model 1 is the champion

3. Hover over the model records to view the various icons.

Figure 39: Mouse over the Model

The icon actions are listed in the following:

a.  Download Model Data.

b.  Open in Pipeline Designer. See the Create Paragraphs using Pipeline Designer section for more details.

c.  Delete Model.

5.3.6 Understand Model Governance

After comparing models, you must understand the Model Governance system in OFS MMG. The Model Governance has an impact on how the application functions with the various user types and the requests they can place from the Model Details window. You require to understand Model Governance before you request model acceptance, review models, or approve models for production.

As discussed earlier, the users are of three types:

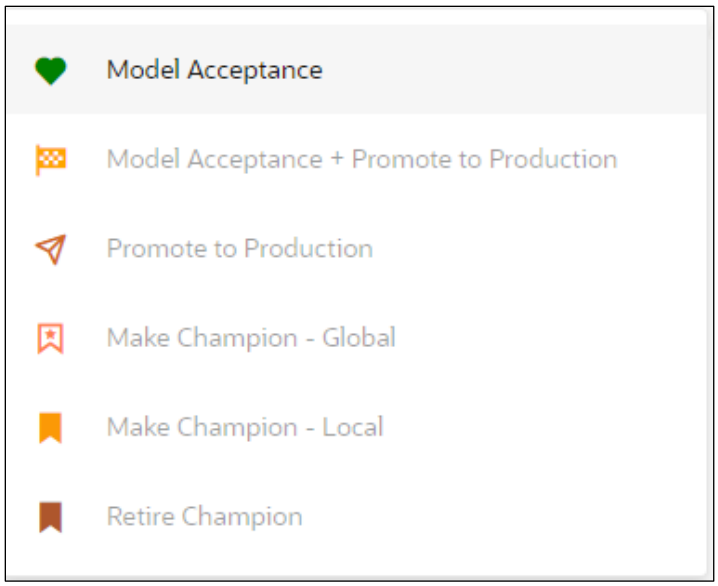
- Requester
- Reviewer
- Approver

The request consists of the following phases in the Request drop-down list (see the Request Model Acceptance section for how to access the drop-down list in the Model Deployment window):

- Model Acceptance
- Model Acceptance + Promote to Production

- Promote to Production
- Make - Champion - Global
- Make - Champion - Local
- Retire Champions

Figure 40: The Request Drop-down List



The values in the drop-down list are active based on the type of user (Requester, Reviewer, or Approver) and the phase that the model is in (accepted, promoted to production, global champion, and so on). Let us look at these with a few examples.

Example 1:

Assume that you are a user with Requester privileges, and you create a model. Now you can request for the model to be accepted on the Model Details window from the **Request** drop-down list. The values enabled for selection are **Model Acceptance** and **Model Acceptance + Promote to Production**. Let us proceed and assume that you select Model Acceptance, then a user with Reviewer privileges forwards your model to a user with Approver privileges. At this stage, the Approver can choose to reject or accept your model acceptance request. A rejection would bring the model back to the initial state with comments on the updates required before it can be requested for acceptance again. However, if the Approver accepts your model, then the **Make Champion - Local** selection is enabled when you log in. You can create many models and send them for acceptance. After acceptance, any model that is accepted can be made the champion on your local workspace at any time replacing the earlier local champion.

Example 2:

Assume that in the previous example, you selected **Model Acceptance + Promote to Production**, then a Reviewer forwards it to the Approver. The Approver, at this stage, chooses to promote the model to production by selecting **Promote to Production**. The model is now available in the production environment and the Approver can choose at any time to select a model from these models in production and select **Make Champion -**

Global. If there exists a Champion model in the production environment, then it will be replaced by the new global champion. However, the earlier champion will still be available in the production environment along with other models and the Approver can choose to make it the global champion again at any time or select any of the other models and make one of them the global champion.

5.3.7 Request Model Acceptance

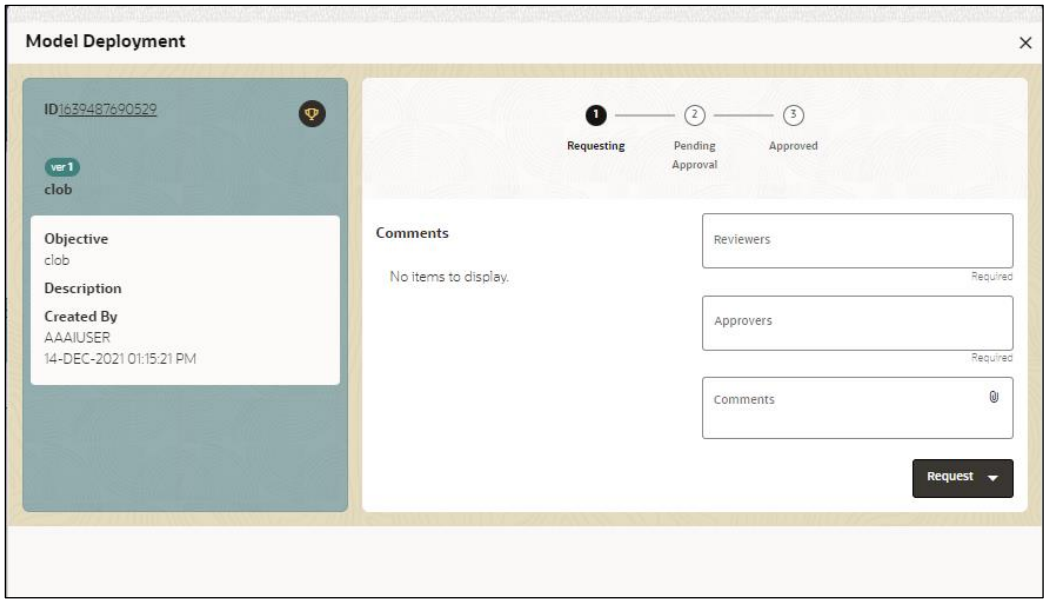
After comparing models, move the selected models to acceptance. Only a user with the Requester role can request for model acceptance. The model will be moved to review which will be available to Reviewer and Approver role, and then to acceptance is available to users with the Approver role, who can promote to production. See the Understand Model Governance section before you start here.

To request a model to promote to production, follow these steps:

1. Click **Model Pipeline**  to display the **Model Pipeline** window.

   The window displays folders that contain models and model records in a table.
2. Click on the Deploy icon on the Model to open it.
3. If a model is a champion, the  icon is displayed on it.
4. To make a model a champion from this window, place the mouse in the selected model columns. and click  to display the **Model Details** window. If a model is a champion model, the  icon is displayed.

Figure 41: Model Details Window



5. Select the Reviewer group from the **Reviewers** drop-down list.
6. Select the Approver group from the **Approvers** drop-down list.
7. Enter comments in **Comments** and click to attach files supporting the comments.
8. Use the following features on the window to perform additional actions.
   - View the model status change in the progress indicator. The Progress Indicator displays the various states of progress that the model has been through. Accordingly, you must request, review, or approve models.

Figure 42: Model Approval Progress Indicator

   - Click the type of request from the drop-down list:
     - Model Acceptance: To review and accept the model creation.
     - Model Acceptance + Promote to Production: To review and promote the model to production.
     - Make Champion - Local: If the model is not the champion model, select to make it the local champion.
     - Promote to Production: To promote a model to production
     - Make Champion - Global: If the model is not the champion model, select to make it the Global champion.
     - Retire Champion: To retire a Champion model
   - Comment History: A record of comments entered in the cycle of model creation and approval with the feature to download attachments.

   The model sent for acceptance or for promotion to production is now displayed to a Reviewer to review it and then to Approver when signed in, who must either accept the request or reject it.

5.3.8 Review Models and Move to Approve or Reject

The Reviewer must provide comments describing the action (approve or reject). If comments are related to rejection and if the Approver rejects, then model goes back to the Requester to make changes or to delete it. If comments are related to approval, then model moves further in the workflow and is displayed to an Approver. See the Understand Model Governance section before you start here.

To review a model, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines**  to display the **Model Pipelines** window.

   The window displays folders that contain models and model records in a table.
3. Click  to display the Model Details Window.
4. Review the details and send it back to the Requester for modifications or send it to an Approver.

5.3.9 Approve Models and Promote to Production

The models reviewed and set to promote to production by either the Requester or Reviewer is displayed to the Approver when signed in. The Approver has to either reject the model and send it back to the requester with supporting comments or approve it for pushing to production. See the Understand Model Governance section before you start here.

> **NOTE** When dataset has used the datasource which is of order (N) for example 5 , and the Production workspace does not contain the datasources at order 5, then promotion of models containing dataset from Sandbox to Production workspace fails.
>
> To remedify this issue, ensure that Sandbox and Production workspace contain the same number of Datasources before you perform promotion of models.

To approve or reject models, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines**  to display the **Model Pipelines** window.

   The window displays folders that contain models and model records in a table.
3. Select a model from the records in the objective.
4. Click  to display the Model Details Window.
5. Click **Approve** or **Reject** with appropriate comments.

5.3.10 Deploy Models in Production and Make it a Global Champion

After approving the models, deploy it to the production environment. You must have an Approver function role and privileges to do this activity.

Note: Sandbox should have the production workspace attached in order to have this option enabled.

To deploy Models in production, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines**  to display the **Model Pipelines** window.

   The window displays folders that contain models and model records in a table.
3. Select the model to deploy and click  to display the details of the model. If a model is a champion, the  icon is displayed.

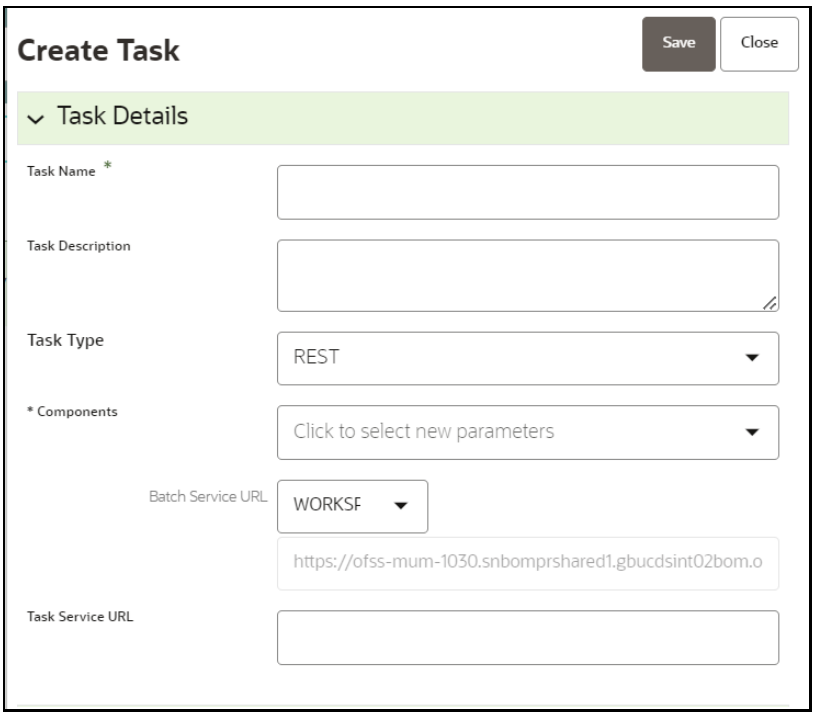
5.4 Execute Models using Scheduler Service

The models that you have created require that they are executed using Scheduler Service before they can be available to the users of OFSAA applications. For more information on this, see the Scheduler Service.

5.4.1 Defining a Task

MMG supports following three Components during the Task creation:

- Model
- Populate Workspace
- Custom



Enter the following details in Task Details section:

- **Task Name:** Enter the task name.

  NOTE: The Task Name must be alphanumeric and should not start with a number. • The Task Name should not exceed 60 characters in length. The Task Name should not contain any special characters except underscore (_).

- **Task Description:** Enter the task description. No special characters are allowed in Task Description. Words like Select From or Delete From (identified as potential SQL injection vulnerable strings) should not be entered in the Description

- **Task Type:** Select the task type from the drop-down list. The options are REST and SCRIPT. You can enter Shell script for Model, Populate Workspace, Custom components. Status key in the curl command should be in uppercase as STATUS.

- **Batch Service URL:** Select the required Batch Service URL from the drop-down list. This can be blank, and you can provide the full URL in the Task Service URL field.

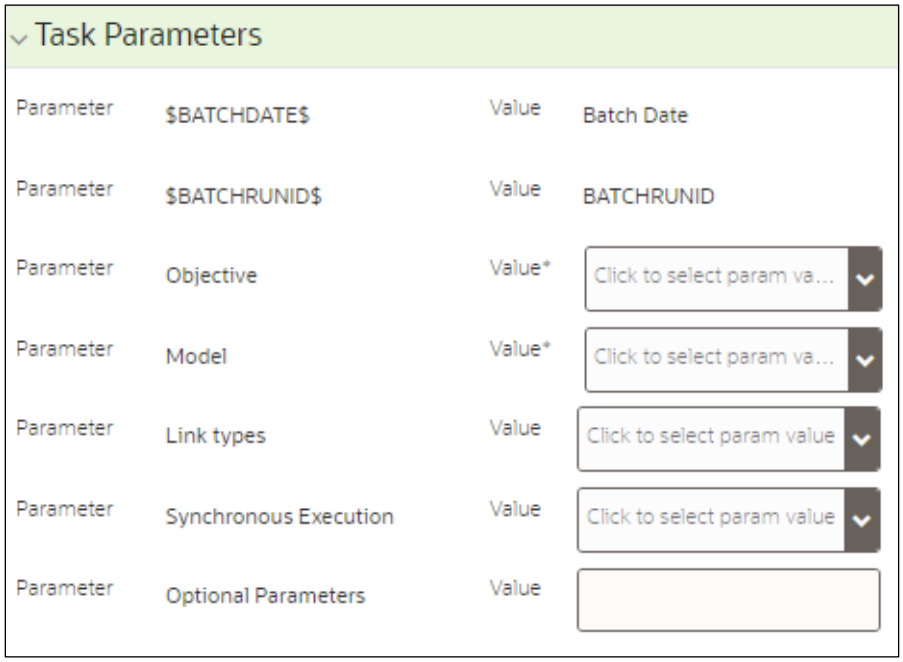
- **Task Service URL**: Enter task service URL if it is different from Batch Service URL.

  NOTE: Task Parameters will vary based on the selected Component.

5.4.2 When Component is Model

The following window shows the Task Parameters for Model Component.

Figure 43: Task Parameters



NOTE: Fields marked with * are mandatory fields.

- **Batch Date:** Shows the batch execution date. You can enable or disable this parameter.
- **Batch Run ID:** Shows the batch execution run ID. You can enable or disable this parameter.
- **Objective:** Select the Object which you want to use for execution. For more information, see the Create Objective (Folders) section. The Sub Objective is displayed with path. For example, if Test1 is Objective and Test11 is Sub Objective, and you want to use Test11 Objective for execution, then select this field as Test1/Test11.
- **Model:** Select the Model of selected Objective. It can be any specific model of Objective or All models of Objective.
  - If the ALL_CHAMPION is selected here, then Objectives with no Champion Model is skipped, and the Objectives with Champion Models gets executed.
  - If CHAMPION is selected, and no Champion is present, then Model Execution gets fail.
- **Link Type**: Select the link type for execution. For example: Training, Scoring, or Training+Scoring. For more information, see the Links in the Pipeline Designer section.
- **Synchronous Execution:** You can set this parameter to Yes or No.
  - If Synchronous Execution is set to Yes, then execution will wait for the notebook execution status.

- If Synchronous Execution is set to No, then execution will not wait for the notebook execution status, it will trigger the notebook and update task status as successful in batch monitor.

- **Optional Parameters:** This is used pass the parameters dynamically.

  For example:
  model_group_name=LOB1,benford_flag=Y,benford_digit=1,from_date=01-Jul-2020,to_date=31-Jul-2021

  Model_group is parameter defined in model and value can be passed here.

The Create Task window also shows the following Header Parameters, which are not editable:

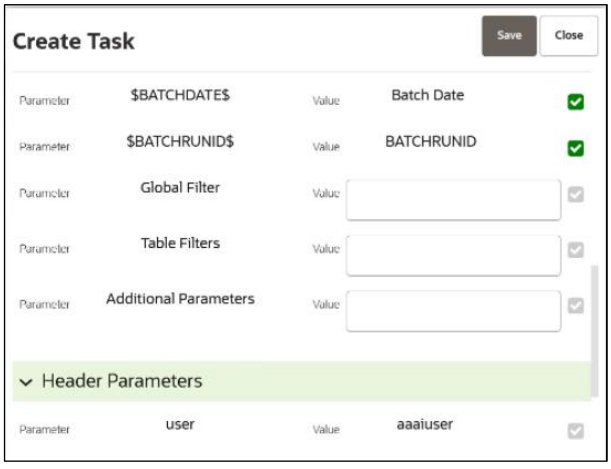
- User: logged in user name
- Workspace: shows the launched workspace name
- Locale: shows the locale. For example: en_US

5.4.3 When Component is Populate Workspace

If you select Component as Populate Workspace, then to use the data population, enter the following parameters

- Additional Parameters
- Table Filters
- Global Filters

For more information, see the Populating Workspace section.



5.5 Import a Workspace Model Data into a New Model

The model data from existing models in OFS MMG in .dmp format and the existing model data in MMG in .dsnb can be imported during the creation of a new model.

Note: The import should happen inside an Objective only.

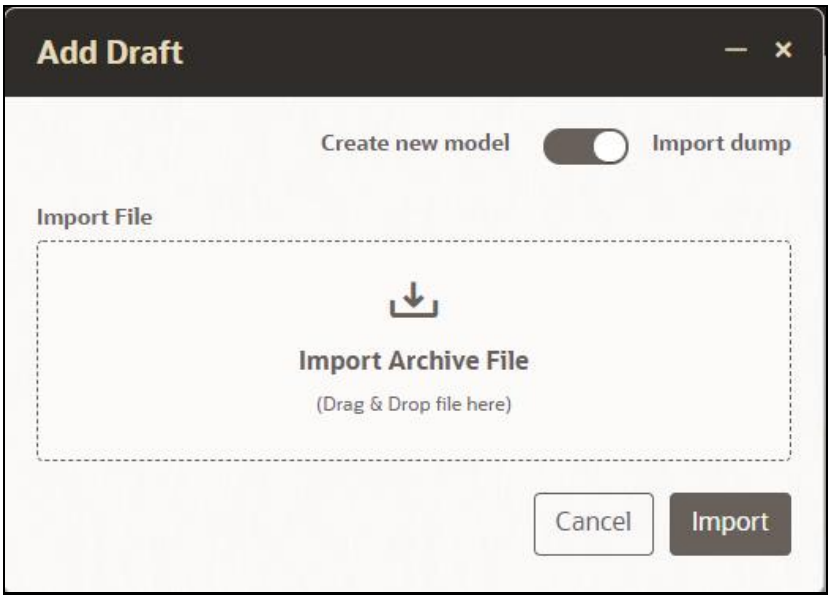
The import of model data lets you reuse and extend on model creation. This topic is part of the procedure of creating Draft Models and after creating a new model using this method, see the Create Draft Models section for instructions on how to proceed further.

To import model data, follow these steps:

1. Click **Launch Workspace** next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines** to display the **Model Pipelines** window.

   The window displays folders that contain models and model records in a table.
3. Select an **Objective**.
4. Double click a model to display the model versions in the expanded display.
5. Hover over a model and click to download the model data dump.
6. Click **Add** and Drafts to display the **Model Details** dialog box for the creation of a new model.

Figure 44: Model Details - Import Dump



7. Drag the toggle switch to select **Import Dump**.
8. Drag and drop the file into the **Import Dump File** field or click in the box to open the file selector dialog and select a file.
9. Click **IMPORT**. A new model is created by importing the model data dump of another model.

5.6 Import / Export Models via Utility

You can import and export the models between Sandbox to Production or vice versa using the Utility.

Prerequisite:

Before you import, ensure the Model artifacts are available in the <installed path>/ftpshare.

To import the models, perform the following steps:

1. Navigate to <MMG_PACK>OFS_MMG/bin and execute in the following format:

   ```
   ./model_export_import_utility.sh IMPORT WORKSPACE LOGIN_USER
   LOCALE FILE_NAME Y/N
   ```

   Example:

   ```
   ./model_export_import_utility.sh IMPORT CS SAUSER en_US
   CS_1638105398036_0.zip Y
   ```

Field	Description
Import	Command to import the Model into Workspace
Workspace	The name of the Workspace to which you are importing the Model
Login_User	The logged in user name.
Locale	The application language preferences For example: en_US
File_name	The file name with the following format: Workspace name_Model ID_Model version.zip For example: `CS_1638105398036_0.zip`

> **NOTE** If you enter input as **Y**, the utility imports the models in the approved status.
>
> If you enter as **N**, the models are imported but not in the approved status.

To export the models, perform the following steps:

Prerequisite:

Before you export, ensure the Models and Drafts are available in the UI / setup.

1. Navigate to <MMG_PACK>OFS_MMG/bin and execute in the following format:

   ```
   ./model_export_import_utility.sh EXPORT WORKSPACE LOGIN_USER
   LOCALE MODELID MODEL_VERSION
   ```

Example:

```
./model_export_import_utility.sh EXPORT CS SAUSER en_US
1638105398036 0
```

Field	Description
Export	Command to export the Model from the Workspace
Workspace	The name of the Workspace from which the model is exported
Login_User	The logged in user name.
Locale	The application language preferences. For example: en_US
Model ID	The Model ID
Model Version	The version of the Model

5.7 Use View Models

The View Models feature launches the OFS MMG Data Studio window. You can view models on this window.

To use View Models, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines**  to display the **Model Pipelines** window.

   The window displays folders that contain models and model records in a table.
3. Click  next to corresponding Model and select Open in Pipeline Designer.

   See the Create Paragraphs in Pipeline Designer sections for details on how to use the OFS MMG Data Studio.

5.8 Edit Models

The editing of models created versions that are different from the previously saved model and the cycle of Model Governance applies to any edited model. You can edit models from the OFS MMG Data Studio window using Python scripting language.

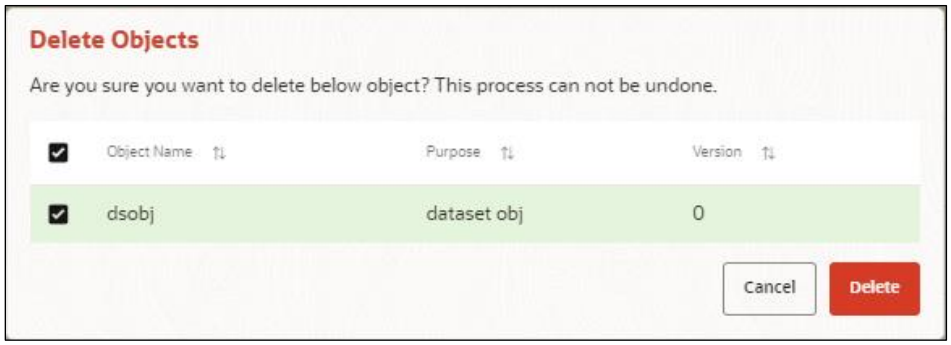
You can edit the script of version 0 only even in Pipeline Designer and Studio. It is not possible in other versions.

5.9 Delete Objectives and Draft Models

To delete Objectives and Models that exist in the Objectives, follow these steps:

1. Click **Launch Workspace** next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Pipelines** to display the **Model Pipelines** window.

   The window displays objectives that contain models and model records in a table.
3. Click next to corresponding record (objective/draft) and select Delete .
4. You can select two or more objectives or models from the records.
5. Click to display the Delete dialog box.
6. Click **Delete** to delete or click **Cancel** to cancel.

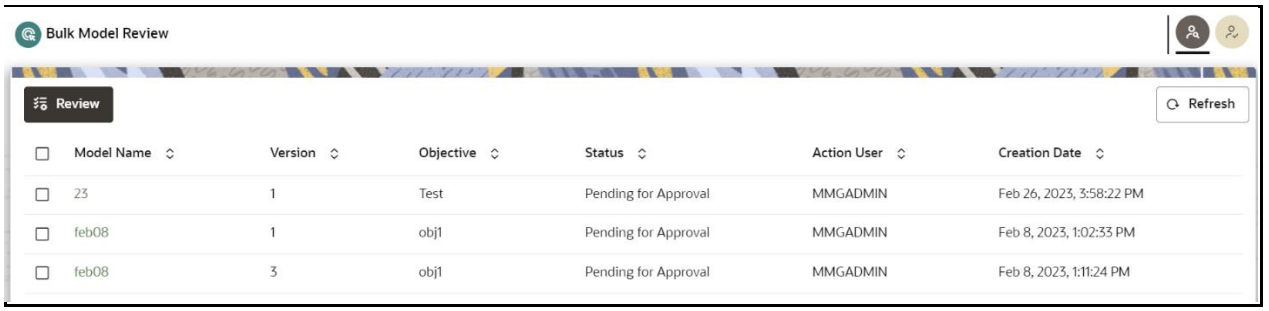


6 Model Actions

The Model Actions window allows you to view the list of models for which the Workflow action has been taken which requires review or approve across the workspace.

For example, when any Model User sends and Model for review or approval, the user receives the bulk Models in Model Actions window for review or approval. This helps when lot of models are received for review before deployment or some actions need to be performed on many models, such as making models champion locally or globally, and so on.

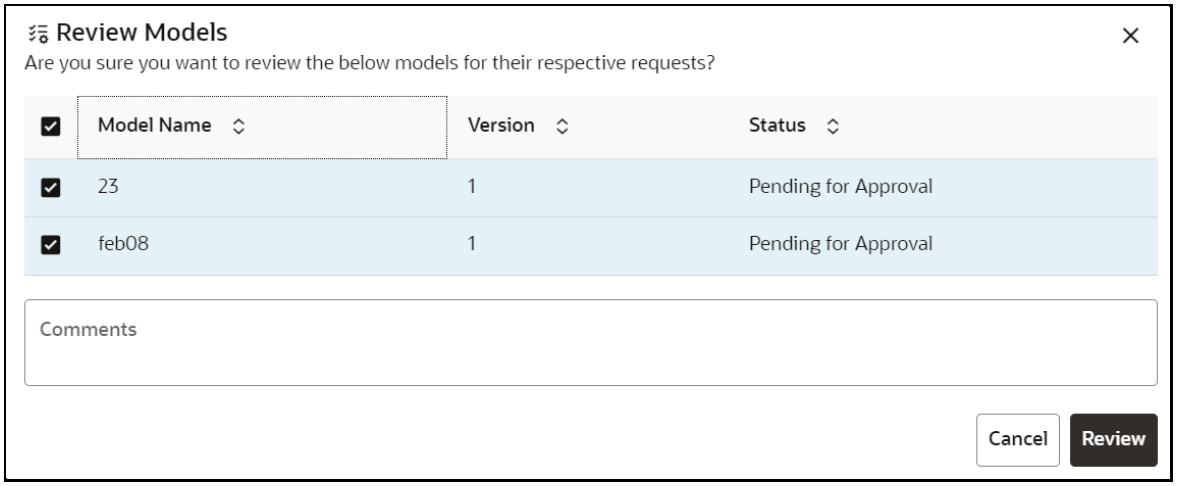
1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Model Actions**  to view the list Models pending for action.

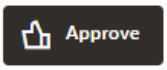


If you are a reviewer, Review Models screen is displayed and if you are an approver, Approve Models screen is displayed. This can be toggled in the header of the screen using Reviewer and Approver icon.

3. To review the model, follow these steps:
   a. Select the Model using the corresponding check box and click Review  .
   b. Enter the comments in Comments field of the Review Models window and click **Review**.

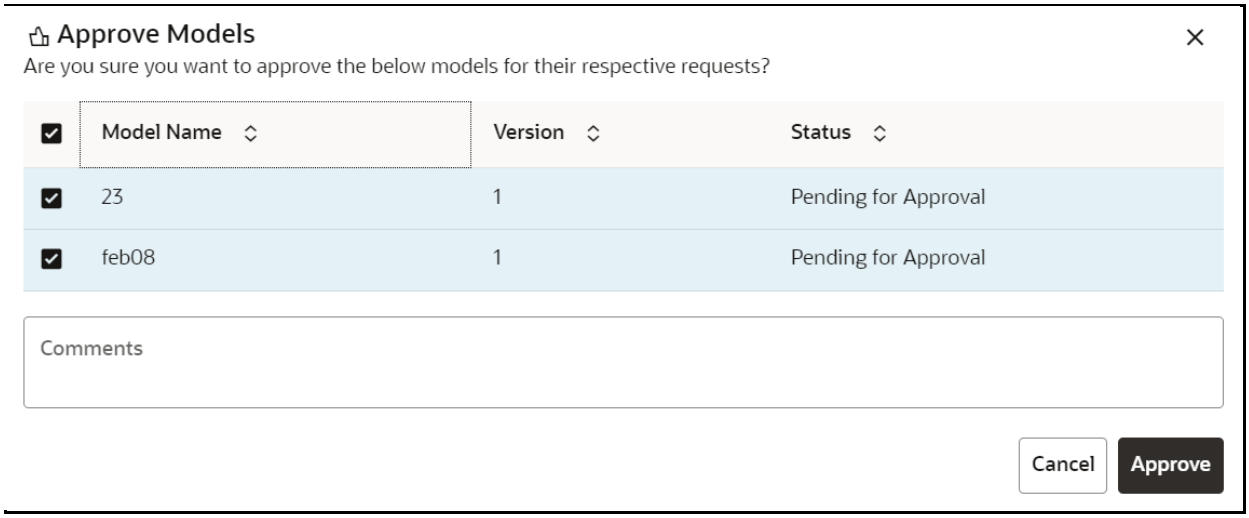
Figure 45: Review Models window



4. To approve or reject the models , follow these steps:
   a. To approve or reject the model, follow these steps: Select the Model using the corresponding check box and click Approve 

or Reject 

.
   b. Enter the comments in Comments field of the Approve Models window or Reject Models window and click Approve or Reject.



7 Graph Pipeline

Unlike traditional relational database management systems, the Graph Pipeline feature allows you to view the data relationships in a graphical format.

This feature uses the latest technology to harness the power of Graph Analytics to give Financial Institutions the ability to monitor the data financial institutions effectively. The data is organized as nodes, relationships, and properties (property data is stored on the nodes or relationships). The results of analytics algorithms are stored as transient properties of nodes and edges in the Graph.

Users can harness the power of Graph Analytics using our in-built in-memory Oracle Graph Analytics Engine (PGX).

The Graph Pipeline functionality allows users to define graphs easily, attach underlying data, and match pipelines to populate data in the graph.

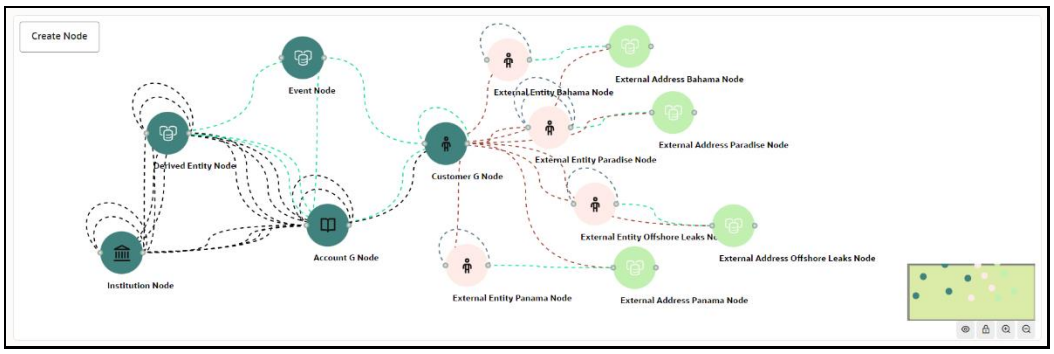
The Graph Pipeline functionality allows users to quickly create and configure a graph for use in advanced analytics. It can also manage and schedule the tasks required to run populate the graph on a periodic basis.

The MMG application allows you to perform the following using Graph Pipeline:

- Creating Graph Model from their existing relational data model
- Configuring the Graph Model through Pipeline UI
- Creating and Scheduling the data pipeline and matching (creation of similarity relationships) for created Graph Model
- Adding new sources and contextualizing the links quickly
- Standardizing the data before pushing it to Graph Model
- Using the following data source for Graph Model:
  - Oracle
  - File System (CSV)
- Using BD data source definitions for pre-seeded Workspace
- Using pre-configured Financial Crime Graph Model pipelines
- Using pre-seeded mappings of Graph Model properties with BD Data source properties
- Scheduling to create Graph by running pre-configured batches
- Blending data and getting insights from data quickly
- Load data into elastic search indexes so matching and entity resolution can occur in the graph
- Define matching rules for the generation of similarity edges in the graph.

The following figure is an example of the preconfigured Financial Crime Graph Model with Nodes and Edges:

Figure 46: Example of Graph Model



The Graph Model defines the nodes and the relationship between them:

- **Node:** It represents a single entity with its attributes. The entity can be a single table or combination of tables merged on some common feature, it could be from a file, or any generalized component of multiple substructures merged for a specific reason/objective.

  For example, Customer.

  A customer could be a single entity.

  Customer, Salesperson, Manager, and so could be individual entities that can be structured together as a “Person” entity with a set of attributes persisted across individual entities.

- **Edge:** Edge is a connector between nodes or the same Node itself. Each edge can have a set of attributes associated with it. These edges will map to 'join' between two entities that could be direct or transitive.

- **Modeler Attributes:** Modeler attributes are the properties of Node/Edge, and these are mandatory inputs.

  For example:

  Customer Node will have properties like Customer ID, Name, Age, Gender, etc.

  A Transaction edge will have properties like Transaction ID, Date, Amount, From/To account, etc.

See the Adding a Graph Pipeline section for creating Graph, Nodes, Edges, and Scheduling

7.1 Accessing the Graph Summary Page

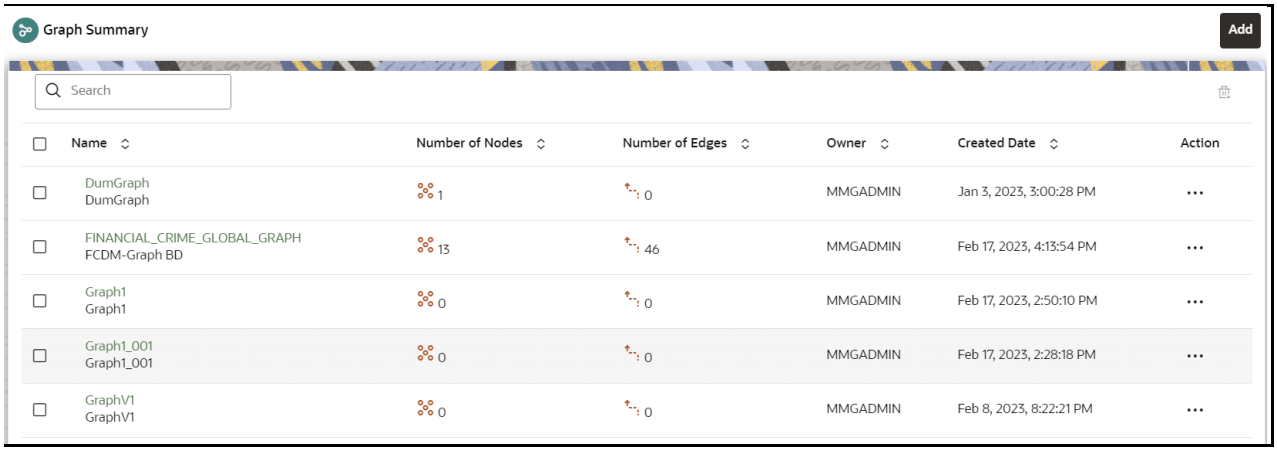
The Graph Summary page gives access to the various graph pipeline functions such as create, view and delete.

To access the Graph Summary page, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click the **Graph Options**  icon .

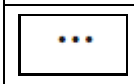
   This displays the graph pipeline in a table.

Figure 47: Graph Summary Page



The following table provides descriptions for the fields and icons on the **Graph Summary** page.

Table 5: Graph Summary Details

Field or Icon	Description
Search	The field to search for a graph pipeline. Enter specific terms in the field for which you want to search, and press Enter on the keyboard to display the results.
Name	The name of the graph pipeline.
Number of Nodes	The number of nodes present in the graph pipeline.
Number of Edges	The number of edges present in the graph pipeline.
Owner	The owner of the graph pipeline.
Created Date	The date on which the graph pipeline was created.
	Click to delete multiple graph pipelines.
	Click to create a new graph pipeline. See Adding a Graph Pipeline section.
	Click **Action** icon to View/Edit/Delete/Download functions on the selected graph pipeline.

7.1.1 Adding a Graph Pipeline

To add a graph pipeline, follow these steps:

1. Navigate to Graph Summary page.

   This page displays the graph summary records in a table.

2. Click **Add**.

   The **Add Graph** pop-up window is displayed.

Figure: Add Graph Window

3. In the **Graph Name** field, enter a name for the graph pipeline.

   The name must be unique to a particular workspace.

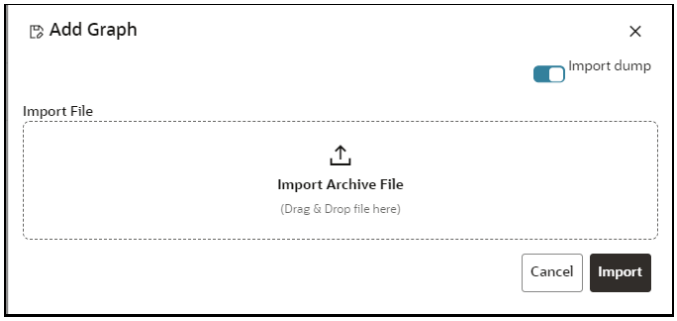
4. In the **Graph Description** field, enter the description for the graph pipeline and click **Add**.

   OR

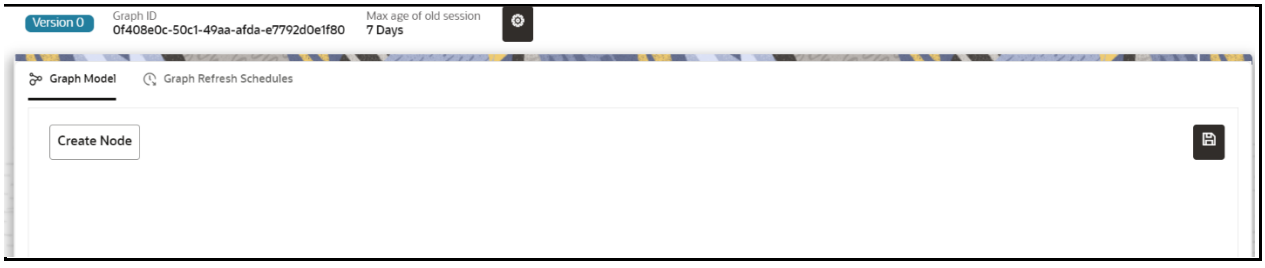
   Use the toggle button to select **Import Dump**.

   The following page is displayed.

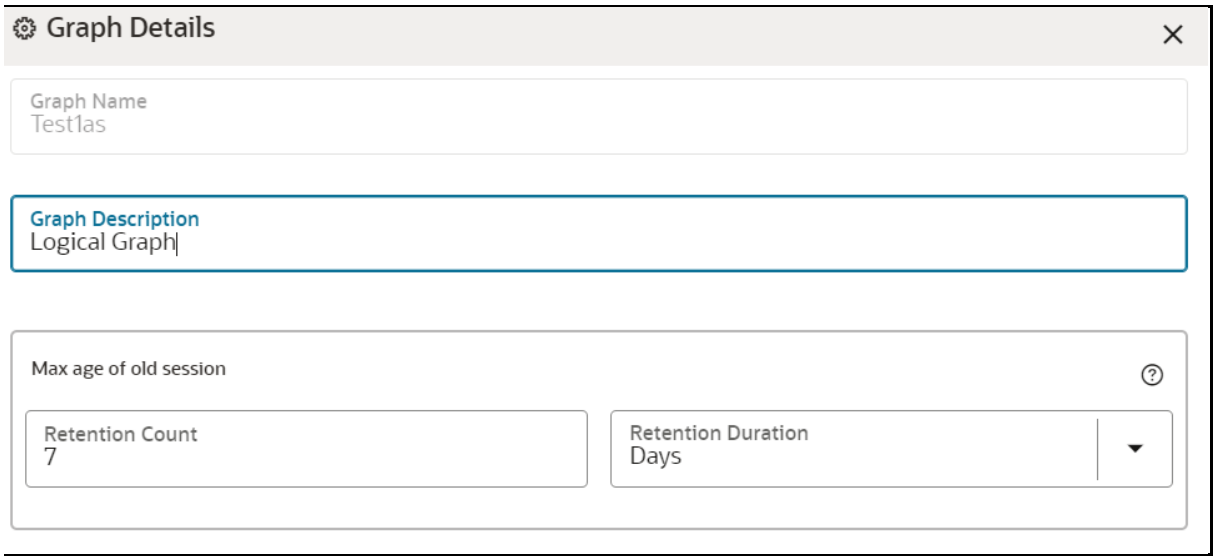
Figure: Import Dump



Browse the file and click **Import**. Once the Import is successful, the Graph Model page is displayed.



The Maximum Age of Old session for a graph is 7 days by default. The Maximum age of old session of 7 days specifies that graph would be retained for a period of 7 days. You can modify the description and Max Age of Old session by clicking on the Setting icon.



Click Save. The user session of the Graph Pipeline will get refreshed after the set timeline.

5. You can perform the following:
    a. Creating a Node
    b. Creating an Edge
    c. Toggle drawing which will enable or disable the user from dragging components
    d. Zoom In and Zoom Out

7.1.2 Creating a Node

To create a Node, follow these steps:

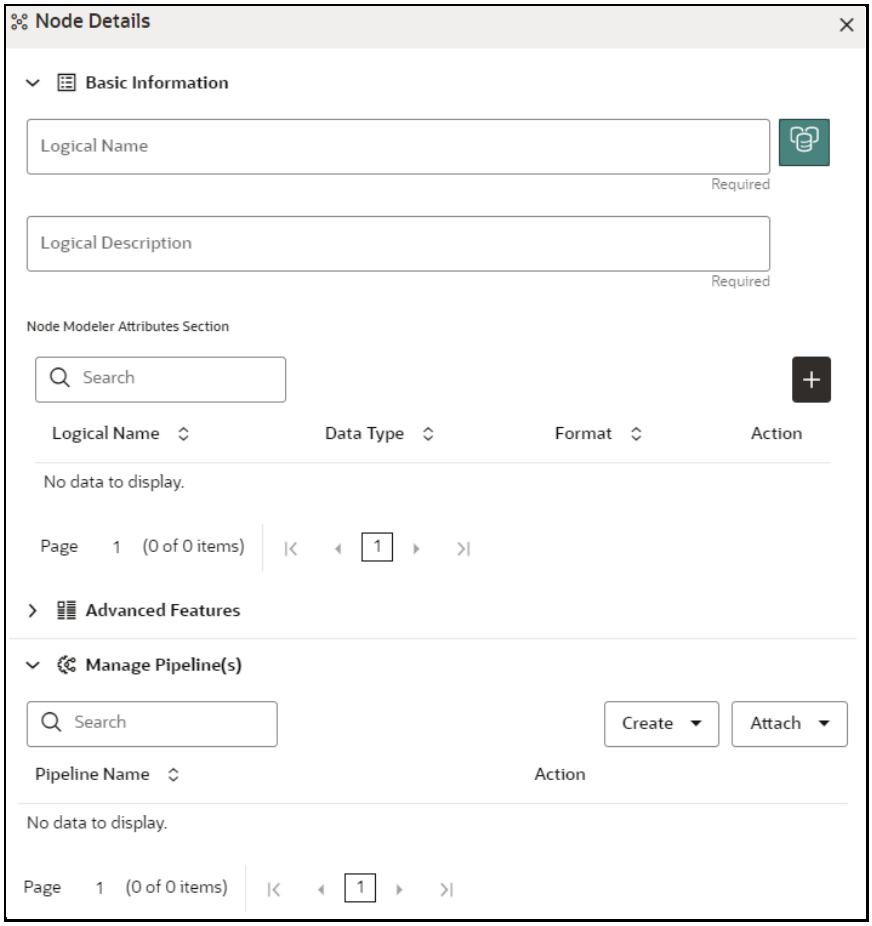
1. In the Graph Model page, click **Create Node**.

    OR

    Select the graph pipeline for which you want to create a Node, click on the **Action** button, and select the **Edit** option. In the Graph Model page, click **Create Node**.

    The **Node Details** screen is displayed.

Figure : Node Details



2. In the **Logical Name** field, enter a name for the Node or click **Setting** icon  .

   The **Setting** pop-up window is displayed.

Figure: Setting

3. Select the required representation from the above Node icon.

   For example: Person, Institution, Account etc.

   Hover over the icon to view the definition of the node.

4. Based on your selection, the related Nodes are displayed. Select the Node, and you can choose the required color for the Node using the **Select Color** option.

5. The selected Node is displayed on the graph pipeline. To add attributes to the Node, click **Add** icon .

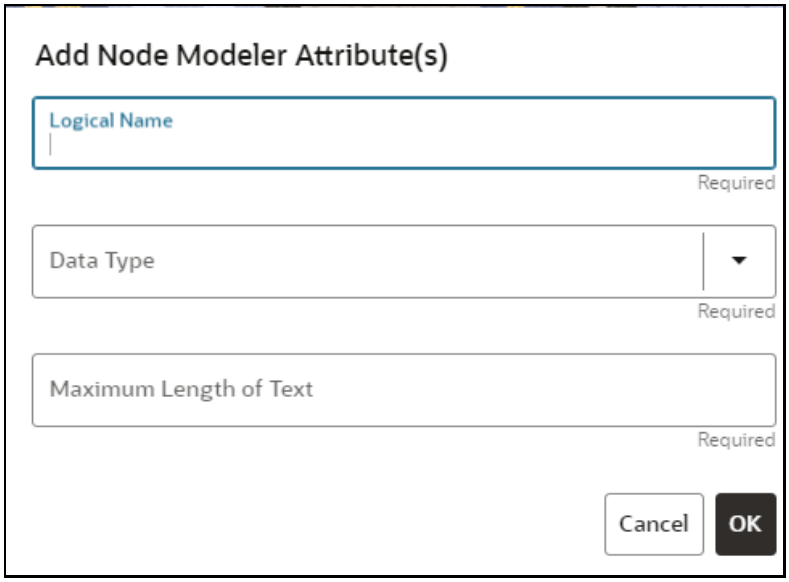
   Modeler attributes are the properties of Node/Edge, and these are mandatory inputs.

   For Example:

   Customer Node will have properties like Customer ID, Name, Age, Gender, etc.

   The **Add Node Modeler Attribute(s)** screen is displayed.

   Figure: Add Node Modeler Attribute(s)



6. In the **Logical Name** field, enter the logical name for the modeler attribute.

   This field is mandatory.

7. Select the data type based on your requirements in the **Data Type** field drop-down.

   The available data types are Text, Number, and Date. This field is mandatory.

   Based on the data type selection, the following input field is displayed.

   For example, the **Maximum Length of Text** field is displayed if the data type is selected as Text.

   > **NOTE** Ensure the Maximum Length attribute is set to at least the field's length from which the data is to be populated.

8. Enter the data and click **OK**.

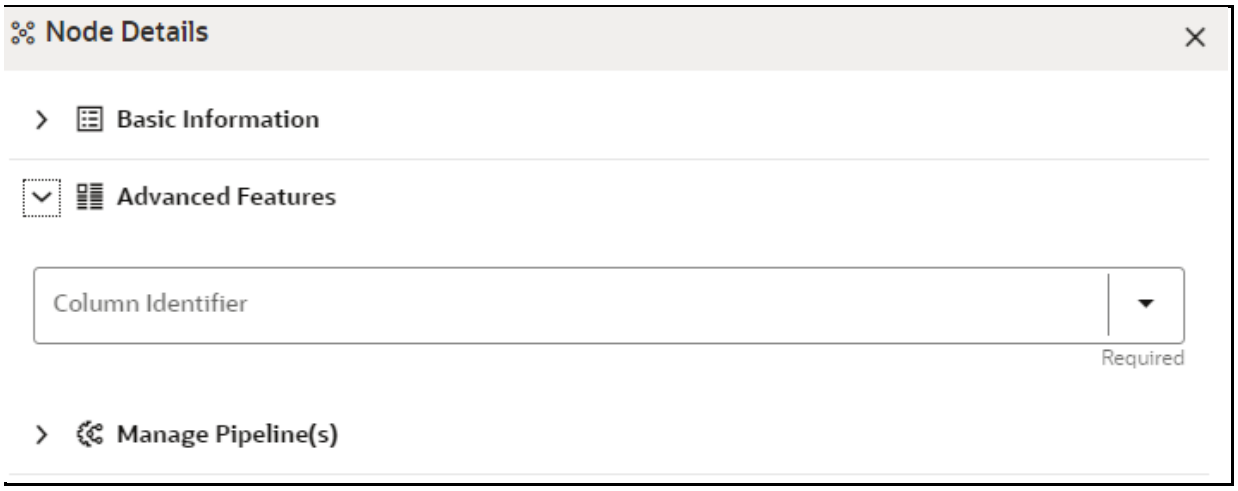
   The Node attributes are displayed on the Node Details page below.

Figure: Search

You can add any number of modeler attributes based on your requirements by clicking on the Add icon.

9. Under the **Advanced Features** group, select the modeler attribute from the Column Identifier drop-down. All the created modeler attributes are displayed under this drop-down.

Figure: Advanced Features



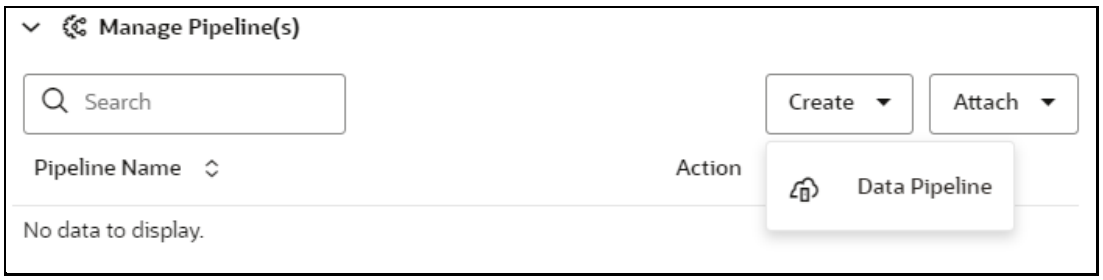
Under the **Manage Pipeline(s)** group, you can either create a new data pipeline or attach the existing pipeline.

The application uses data pipelines to prepare filtered data which can be used to create and run scenarios. Data pipelines prepare data by selecting and joining data sources to create virtual data tables, adding derived attributes to data, running derivations on the data to determine the risk associated with the entity, and so on.

> **NOTE** The newly created data pipeline is not automatically added to the Node. You must attach the pipeline using the **Attach** drop-down option.

The selected data pipeline is displayed under the Manage Pipeline(s) group.

Figure: Manage Pipeline(s)



10. Click **Create** drop-down and select **Data Pipeline**.

    The Create Pipeline window is displayed.

Figure: Create Pipeline

11. Select the **Create New Data Pipeline** option and then click **Ok**.

    The **Pipeline Designer** page is displayed.

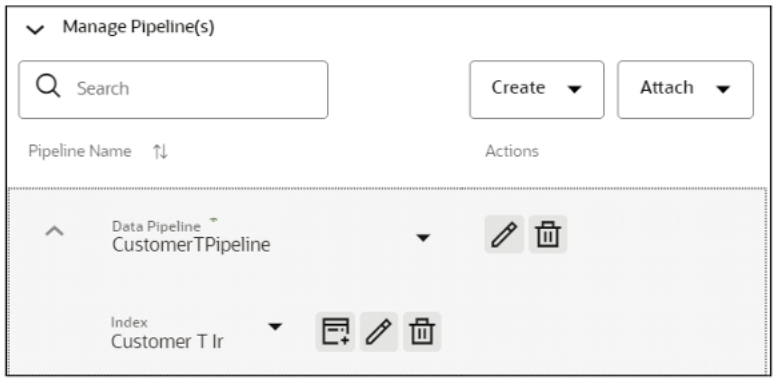
    - To create a data pipeline, see the Managing Data Pipeline section.

      OR

    - To clone from an existing data pipeline, see the Cloning from an Existing Data Pipeline section.

> **NOTE** The Persist of the Data pipeline of the corresponding node should be defined with the prescript. see the Prescript Condition section for more details.

Figure: Create an Index

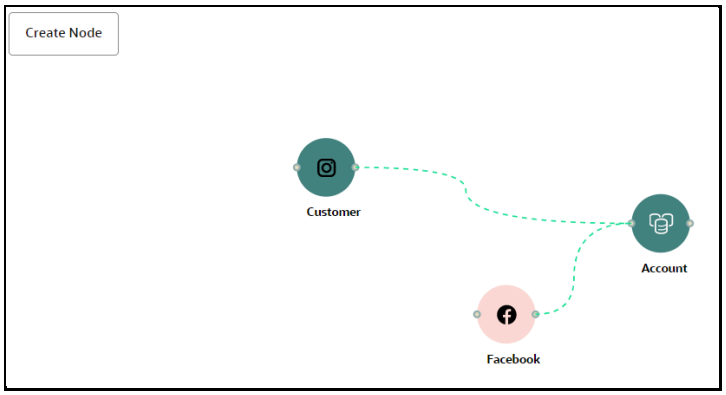


- To define an index for the data pipeline, see the Defining an Index section.

12. By clicking on the **Attach** drop-down, you can add any number of data pipelines to the Node-based on your requirements.
13. Click **OK**.

    The Node is created with pipeline Facebook and an account attached to it.

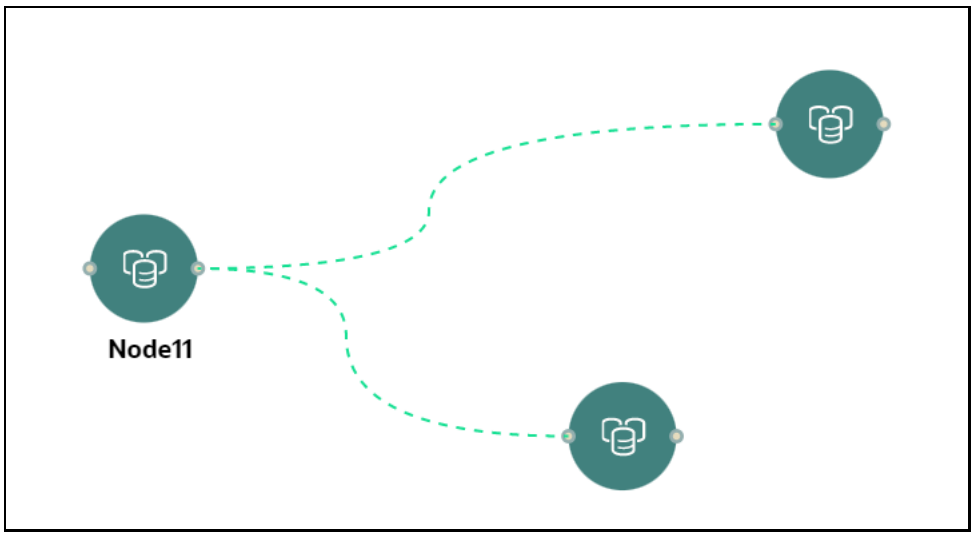
Figure 48: Graph with Node and Edge



7.1.3 Creating an Edge

An Edge can be used to connect two different nodes or connect to itself. When compared to nodes, edges also capture the basic information like name, description, attributes to an edge, and possible pipelines that can be attached to an edge.

Figure: Edge

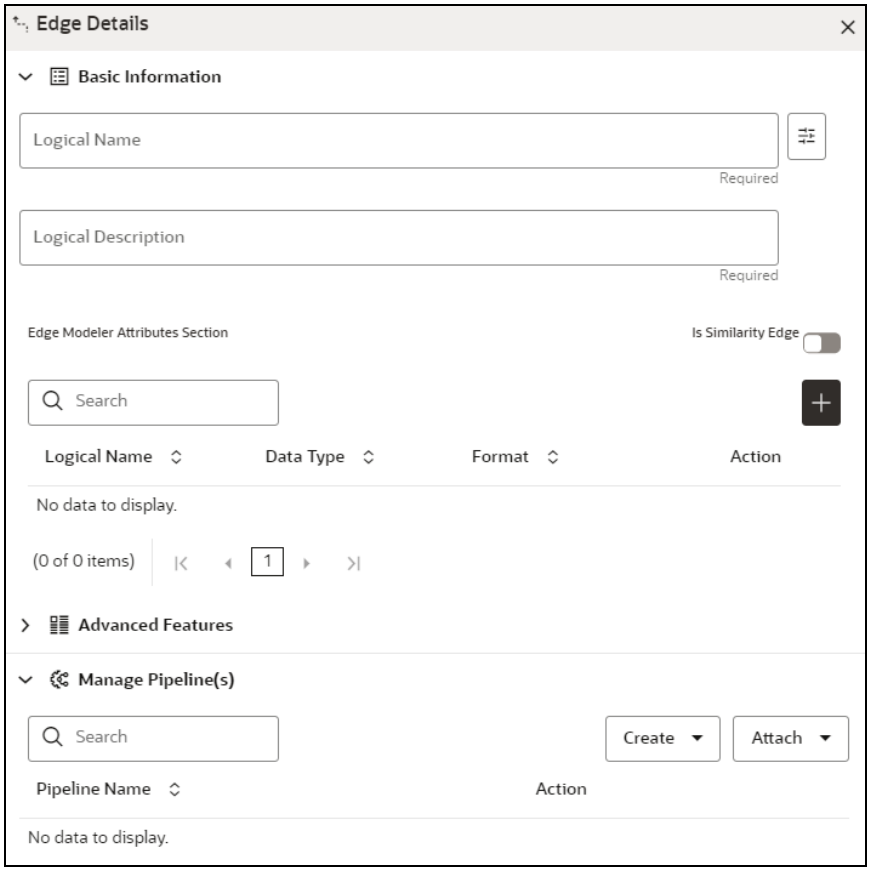


To create an edge, perform these steps:

1. Hover on the Node to create a relationship between them.

   The **Edge Details** screen is displayed.

Figure: Edge Details



2. In the **Logical Name** field, enter the Node's name or click the Setting icon  .

   The **Setting** pop-up window is displayed.

Figure: Setting

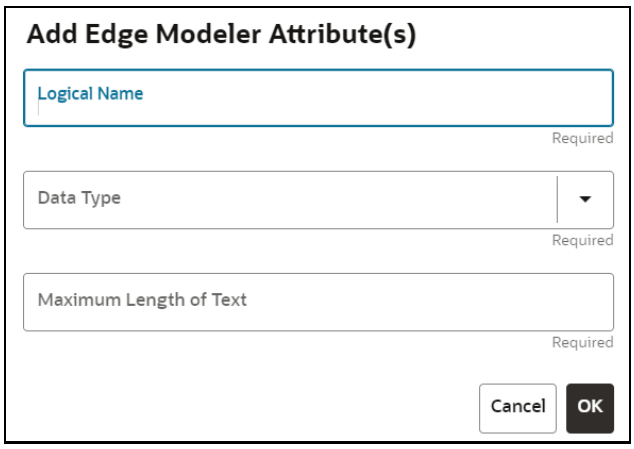
3. Select the required edge from the options displayed and choose the required color for the edge using the **Select Color** option.
4. Click **OK**.

   The selected edge is displayed on the graph pipeline

5. To add attributes to the edge, click **Add** icon  .

   The **Add Edge Modeler Attribute(s)** screen is displayed.

Figure: Add Edge Modeler Attribute(s)



6. In the **Logical Name** field, enter the logical name for the edge attribute.

   This field is mandatory.

7. In the **Data Type** drop-down, select the data type based on your requirements.

   The available data types are Text, Number, and Date. This field is mandatory.

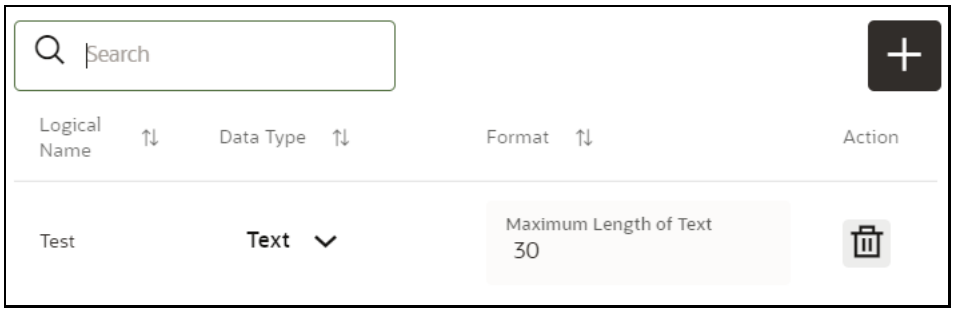
   Based on the data type selection, the following input field is displayed.

   For example, the **Maximum Length of Text** field is displayed if the data type is selected as **Text**.

8. Enter the data and click **Ok**.

   The edge attribute is displayed on the Edge Details page.

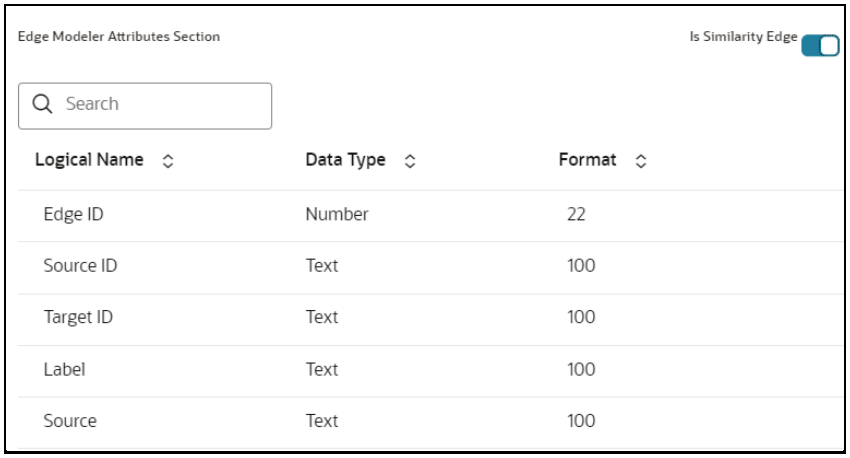
Figure: Search



By clicking on the **Add** icon, you can add any number of edge attributes based on your requirements.

OR

Figure: Edge Details



Drag the toggle button to **Is Similarity Edge** in the **Edge Modeler Attributes Section** to add the pre-defined set of attributes available in the edge.

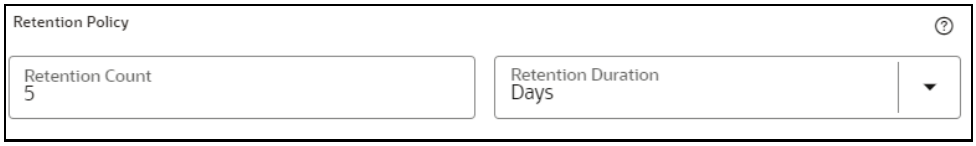
Similarity edges in the graph are created by Match Rules. If the edge is defined using Match Rule (under Manage Pipeline, by selecting Match Rule from drop down), the toggle button should be enabled to make it a Similarity Edge.

9. Under the **Advanced Features** group, you can specify the Edge Retention Policy.

   For example, An Edge Retention Policy of 7 Days specifies that the current edge would be retained in the physicalized Graph for a period of 7 Days.

10. Enter the retention count and select the retention duration from the drop-down.

    The available retention durations are Days, Weeks, and Months.

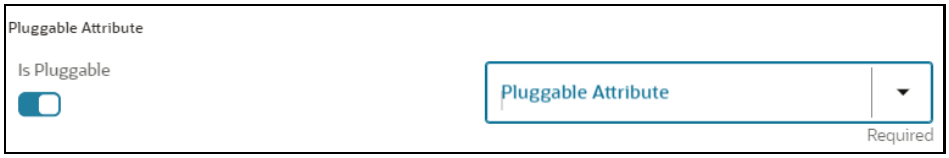


11. Enable the **Pluggable Attribute**, then the corresponding drop-down will contain the list of attributes of the Edge.

    Pluggable edges are those edges whose data can be directly plugged in to the graph. In Financial Crime and Compliance domain, Transaction is a pluggable edge, which is generally large and is difficult to find out difference/ delta between two subsequent loads. Hence, it's data is provided for a range of dates which can be directly plugged in to the PGX memory while refreshing the graph.

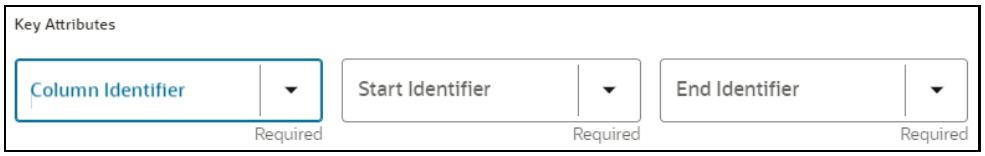
    Any pluggable edge should have an identifier attribute, which is of "Date" data type.

Figure: Pluggable Attribute



12. Under the **Key Attributes** field, select the Column, Start, and End identifier from the respective Identifier drop-down. Without these attributes, an edge cannot be created.

All the created edge attributes are displayed under this drop-down.



13. Under the **Manage Pipeline(s)** group, you can either create a new data pipeline if the edge is coming from source data and Match Rules, if a similarity edge is to be created between nodes or attach the existing pipeline and Match Rules.

Figure: Manage Pipeline(s) for Edge

14. Click Create drop-down and then select the Data Pipeline.

The Create Pipeline window is displayed.

Figure: Create Pipeline

15. Select the **Create New Data Pipeline** option and then click **OK**.

The Pipeline Designer page is displayed.

- To create a data pipeline, see the Creating a New Data Pipeline section.

  OR

- To clone from an existing data pipeline, see the Cloning from an Existing Data Pipeline section.

> **NOTE** The Persist of the Data pipeline of the corresponding edge should be defined with the prescript. See the Prescript Condition section for more details.

16. Click **Create** drop-down and then select the **Match Rules** from the Manage Pipeline(s) group.

The **Create Pipeline** window is displayed.

Figure: Create Pipeline

17. Select the Create New Match Rule option and then click Ok.

    The Create Match Ruleset window is displayed

    - To create a Match Rule, see the Creating Match Ruleset section.

      OR

    - To clone from an existing match rule, see the Cloning Ruleset (Match) section.

> **NOTE** The newly created data pipeline and match rule are not automatically added to the edge. You must attach using the Attach drop-down option.

The selected data pipeline/match rule is displayed.

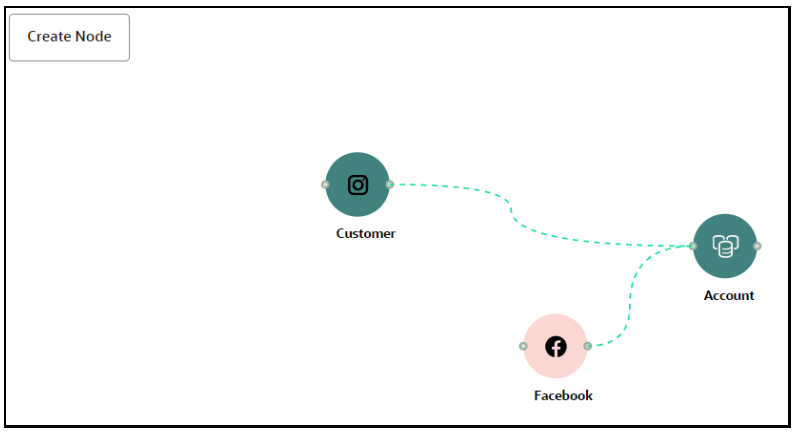
Figure: Create Pipeline

By clicking on the **Attach** drop-down, you can add any number of data pipelines/match rules to the Node-based on your requirements.

18. Click **OK**.

    An Edge is created between the Customer, Facebook, and Account.

Figure 49: Graph page



7.1.4 Zoom In and Zoom Out

You can zoom in or zoom out of the graph pipeline and view the data pipelines using the diagram present in the bottom right corner of the page.

- Click to zoom in on the graph pipeline.
- Click to zoom out the graph pipeline.
- Click to view the data pipelines.

7.1.5 Editing a Graph Pipeline

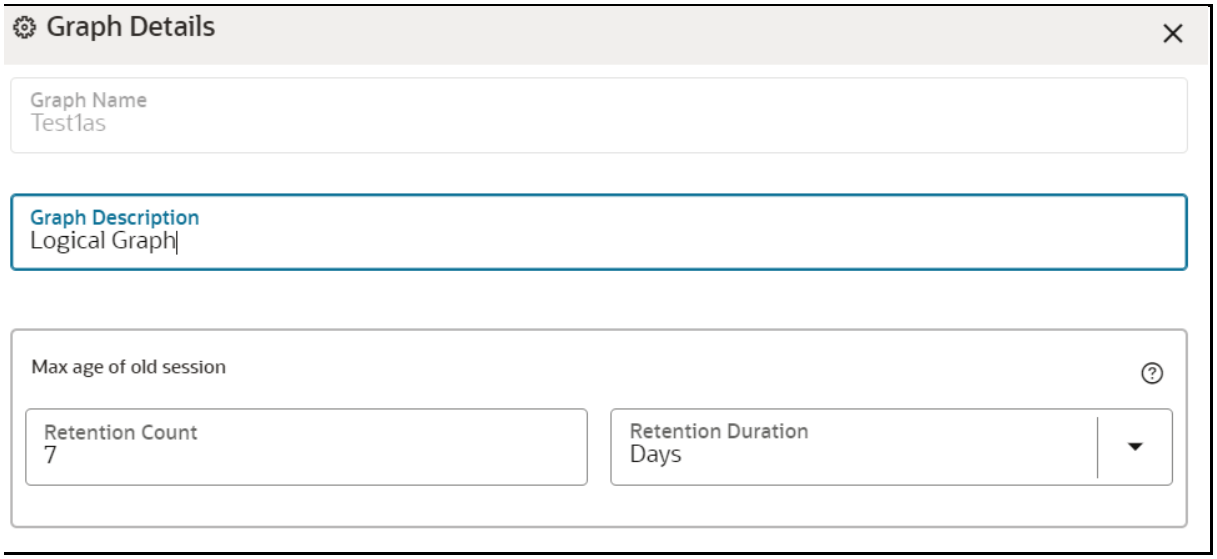
To edit a graph pipeline, perform these steps.

1. In the **Graph Summary** page, select the graph pipeline you want to edit, click the **Action** icon, and select **Edit**.
2. In the **Graph Name** field, modify the graph pipeline name.

   The name must be unique to a particular workspace.
3. Click Setting icon
4. In the **Graph Description** field, enter the description for the graph pipeline and click **Save**.

   The following page is displayed.

Figure: Graph page



The Maximum Age of Old session for a graph is 7 days by default. The Maximum age of old session of 7 days specifies that graph would be retained for a period of 7 days. You can modify the description and Max Age of Old session.

5. Click **Save**.

   The user session of the Graph Pipeline will get refreshed after the set timeline.

7.1.6 Deleting a Graph Pipeline

To delete a graph pipeline, perform these steps.

1. On the **Graph Summary** page, select the graph pipeline you want to delete, click the **Action** icon, and select **Delete**.

   To delete multiple graph pipelines, select the graph pipelines which you want to delete and click  Delete icon in the Header..

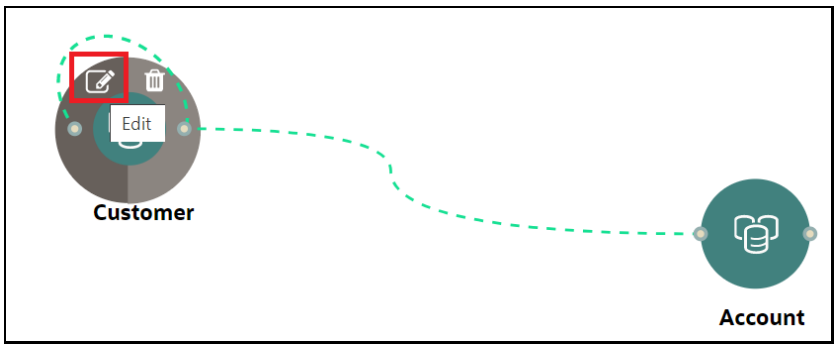
7.1.7 Editing a Node

To edit a node, perform these steps.

1. On the **Graph Summary** page, select the graph pipeline you want to edit, click the **Action** icon, and select **Edit**.
2. Hover over the graph pipeline and click **Edit** icon.

   The **Node Details** page is displayed.

Figure: Edit Node



3. Perform the steps from 3 to 12 in the Creating a Node section.

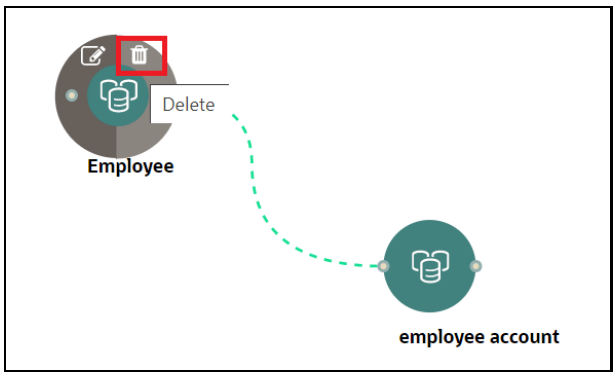
> **NOTE** If you delete any attribute in the Node Modeler Attribute section of the Node Details page, you must delete the same attribute in the match rule details.
>
> For example:
>
> If the edge has two match rules and one is attached, and another match rule is not attached to the edge details. You need to remove attributes in both math rules for this case. To remove the deleted attribute from the Node Details page in the unattached match rule, perform the following:
>
> 1. Navigate to **Edge Details > Manage Pipeline(s) > Attach**. Attach the new match rule, which is not attached to the node details.
> 2. Click **Edit** icon and delete the attribute in the Ruleset Details, which is removed from the Node Details page.
> 3. Click **Save**. The attribute is removed.
> 4. Click **Delete** in Manage Pipeline(s) of the Edge Details page to remove the attached match rule.
> 5. After saving in the Edge Details page, navigate to the Node Details page and delete the attribute again.
> 6. Click **OK**. The attribute is removed from the node.

7.1.8 Deleting a Node

To delete a node, perform these steps.

1. On the **Graph Summary** page, select the graph pipeline you want to edit, click the **Action** icon, and select **Edit**.
2. Hover over the graph pipeline and click **Delete** icon.

Figure: Delete Node



A confirmation dialog is displayed.

3. Click **Delete**.

   The Node is deleted.

7.1.9 Creating Graph Refresh Schedules

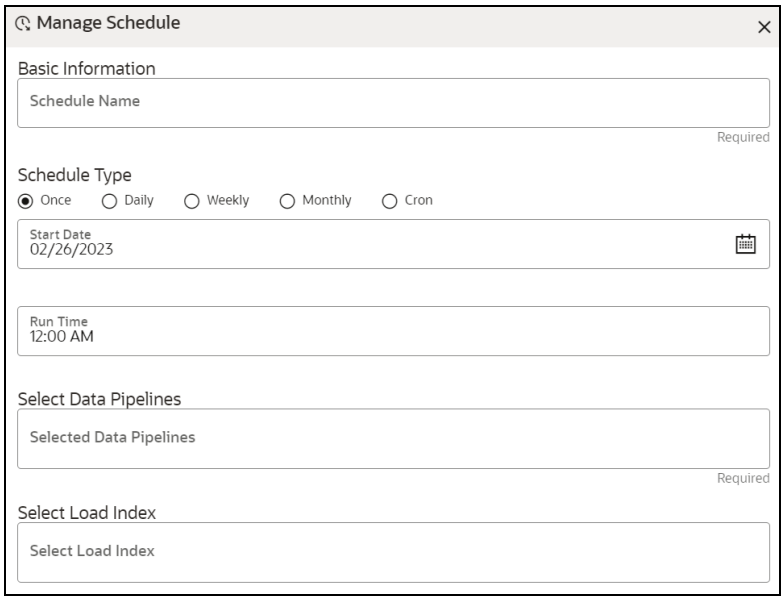
> **NOTE** Currently, when you execute a Graph pipeline for the first time and navigates to the "Monitor Batch" screen, two "run ids" are generated for that batch.

To create a Graph Refresh Schedule, perform these steps.

1. In the **Graph Refresh Schedules** page, click **Add Schedule**.

   The **Manage Schedule** screen is displayed.

Figure: Add Schedule



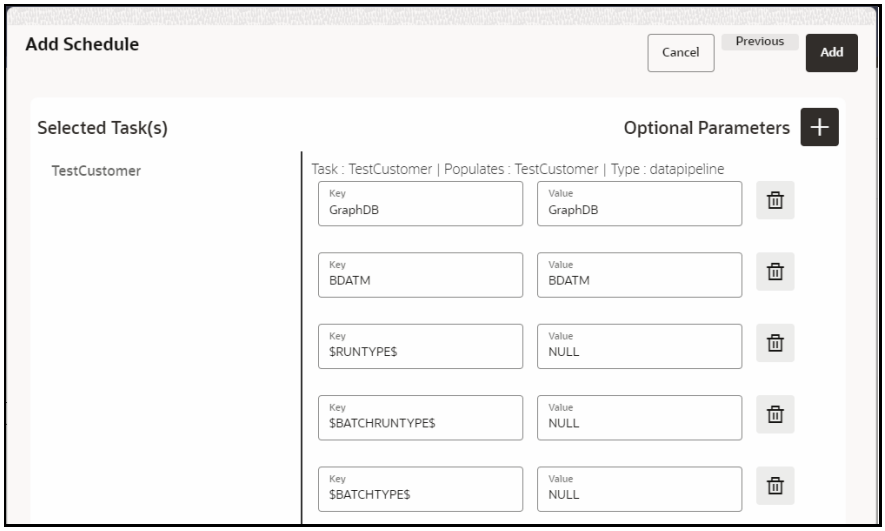
2. In the **Schedule Name**, enter the desired name.
3. Select the **Schedule** Type from the options.

   Based on the schedule type, the following input field is displayed.

> **NOTE** For more details and expressions of cron schedule type, click the **help** icon.

4. Click inside the **Select Task** field and select the tasks. You can select multiple tasks.
5. Click **Next** and add the values for the selected task, and the values are mandatory fields.

Figure: Add Optional Parameters



6. Click + icon to add optional parameters for the selected task.

   For the Pre-configured Data pipeline, respective Key fields will be displayed. You can configure the respective key-value in the Value field as required and specified in the following table:

Table: Optional Parameters for Data Pipeline

Name	Description	Default Value
$GRAPH_SCHEMA	Oracle wallet alias of graph schema. Navigate to <OFS_COMPLIANCE_STUDIO>/wallet/ tnsnames.ora and check the Graph Schema alias name. For example, graph_schema_alias	
$BD_SCHEMA	Oracle wallet alias of BD schema. Navigate to <OFS_COMPLIANCE_STUDIO>/wallet/ tnsnames.ora and check the BD Schema alias name. For example, bd_schema_alias	
$RUNTYPE$	Indicates Run type. It can be either a production batch or a test batch. If it is a production batch, set it as PROD and TEST for the test batch.	PROD
$batchRunType$	Indicates Batch Run type. It can be either run or re-run. If it needs to execute once, set it as RUN and RE-RUN for re-executing after failure.	RUN
$BATCHTYPE$	Indicates the batch type. It should be DATA for the data pipeline task.	DATA
$JOBNAME$	Name of the data pipeline to be executed.	

7. To add Match Rules, enable the Process Matching Rule option and click **+** icon to add optional parameters.

   For more information on Match Rule, see the Creating Match Ruleset section.

Figure: Process Matching Rule

For the Pre-configured Match Rules, respective Key fields will be displayed. You can configure the respective key-value in the Value field as required and specified in the following table:

Table: Optional Parameter for Match Rules

Name	Description	Default Value
datasource	Oracle wallet alias of graph schema. Navigate to <OFS_COMPLIANCE_STUDIO>/wallet/ tnsnames.ora and check the Graph Schema alias name. For example, graph_schema_alias	-
loadType	Execution Load Type. Available types are full and delta. In this release, only the full mode is supported.	full
processingGroupName	Logical processing group name for the batch. You should set it as MAN.	MAN
basedOnPipelineId	Match rules can be executed based on pipeline Ids or ruleset Ids. The value should be true while running from the Graph.	true
runSkey	A numeric value indicates the runskey of the execution.	100
runType	Indicates matching job that can be executed as part of graph or Entity Resolution. It should be set as Graph.	graph

8. Enable the **Refresh Graph** option and click **Add** icon to refresh the Graph to add optional parameters.

> **NOTE** The Key and Value parameters are not required.

Figure: Refresh Graph

> **NOTE** If you do not enable the Refresh Graph option, the pipeline will add nodes and edges to the graph schema but does not load into PGX.

8 Scheduler Service

The Scheduler Service automates the running of models in the Compliance Studio application.

The Scheduler Service contains a graphical user interface and a single control point for defining and monitoring background executions.

The Scheduler Service is a service in the Infrastructure system that automates behind-the-scenes work that is necessary to sustain various enterprise applications and functionalities. This automation helps the applications to control unattended background jobs program execution.

The Scheduler Service contains a graphical user interface and a single point of control for the definition and monitoring of background executions.

Following are the concepts or terminologies in the Scheduler Service:

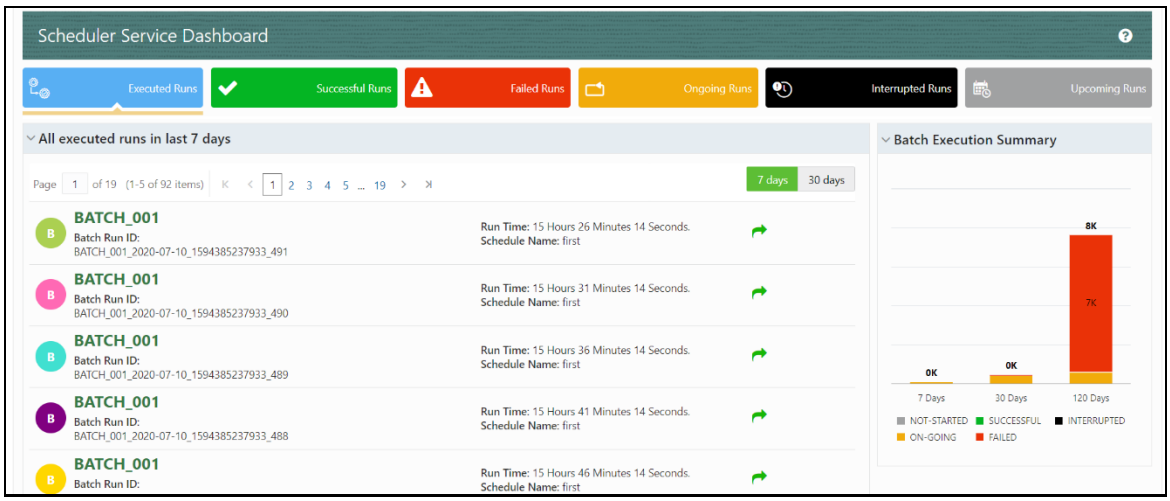
- **Batch**: Date and time-based execution of the background tasks based on a defined period during which the resources were available for batch processing.
- **Job**: A batch job is a piece of a program meant to meet specific and business-critical functions. The program is a RESTful API used in a batch.
- **Job Dependency**: When the batch job is submitted, it is moved to the job queue until the system is ready to process. The system process the job based on chronological order or priority in case if more jobs are required to be executed in the job queue.
- **Schedule**: Batch jobs are used to automate the tasks that require to be performed on a regular basis but don't necessarily need to occur during the day or have an employee interacted with the system are batch schedule. Jobs that happen on a regular basis are incorporated into batch schedules.

Monitor: The Scheduler Service enables you to monitor your executions by using a web-browser. It provides real-time feedback on the status of the current encoding job and lists the jobs pending in the batch. You can Cancel or Restart the service when required.

8.1 Scheduler Service Dashboard

On accessing Scheduler Service using your browser, the **Scheduler Service Dashboard** window is displayed.

Figure 50: Scheduler Service Dashboard

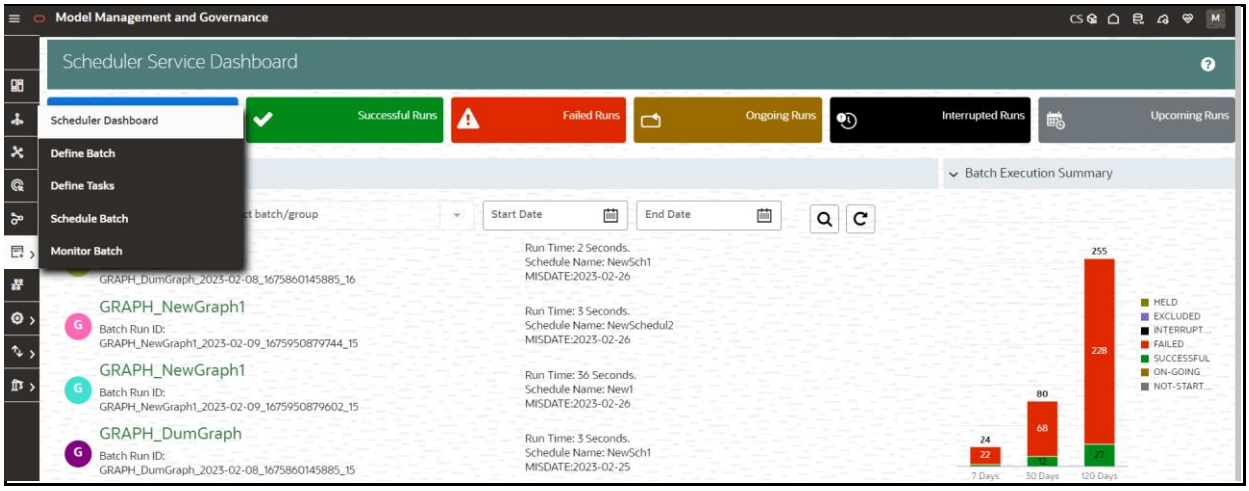


8.2 Header Details

The Header Details in the **Scheduler Service** window is displayed as shown and it has the following options.:

- Menu Navigation: Shows the menu options available in Scheduler Service.
- Quick Actions: Quick actions can be performed by using the buttons in this section.
- User Details: The User details are displayed, you can view the help, set you profile preferences.

Figure 51: Header Details



On the left side of the window, click the **Scheduler Service** icon to see the different option in the Scheduler Service.

8.3 Scheduler Service Dashboard

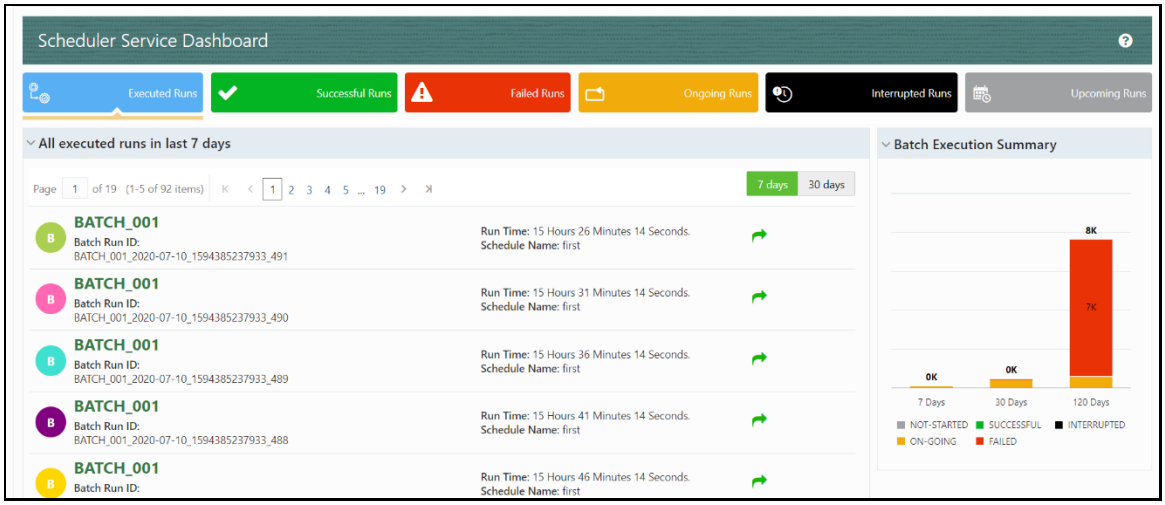
To access the Scheduler, In the OFSAA Applications window, select the Menu Navigation icon, and then select Batch Administration, and then select Scheduler to open the *Dashboard* is displayed.

To view the demonstration of the Dashboard window, see the *Scheduler Service Introduction* video.

Scheduler Service Introducing Video

In the Scheduler Service window, you can view the following details:

- The *Executed Runs*, *Successful Runs*, *Failed Runs*, *Ongoing Runs*, *Interrupted Runs*, and *Upcoming Runs tabs*. You can click the tabs to view the details of the Batches based on their status. For example, click **Ongoing Runs** to view the details of the batches that are currently running.
- The Batches that were executed within the last 7 or 30 days contain details such as Batch Name, Batch Run ID, and Run Time. Click **30 days** to view the batches that were executed within the last 30 days. You can click the icon corresponding to a Batch to monitor it.
- The **Batch execution Summary** pane displays the count of total batches executed that were executed within the last 7 days, 30 days, and 120 days. Additionally, you can see the separate count of successful batches, failed batches, interrupted batches, on-going batches, and the batches which are yet to start, by hovering your mouse the batches.



8.4 Define Batch

The **Define Batch** window displays the details of all existing Batch like Batch ID, Batch Name, Batch Description, Last Modified By and Last Modified Date. This window allows you to create a new, edit, copy, and delete the batches. You can also create a Batch Group and execute the Batch Group which has the list of batches that you have selected and grouped.

To navigate to the Define Batch window, click Define Batch option from the Header in the Dashboard window. After selecting the batch, you can select the icon corresponding to the batch to proceed to create or edit tasks.

8.4.1 Batch

Batch is a process of execution of the Date and time-based background tasks based on a defined period during which the resources were available for batch processing.

8.4.1.1 Create a Batch

You can create a new batch in the Define Batch Window and schedule and monitor the batch that you created.

To create a new Batch, perform the following steps:

1. In the Define Batch window, click  .

   The **Create a New Batch** window is displayed.

2. Specify the details as tabulated in the Create a New Batch window as described in the following table.

   Table 1: Fields in the Create a New Batch window and their Description

Field	Description
Batch Details	
Batch Name	The Batch Name is generated based on the values provided by you. **NOTE:** • The Batch Name should be unique across the Information Domain. • The Batch Name must be alphanumeric and should not start with a number. • The Batch Name should not exceed 60 characters in length. • The Batch Name should not contain any special characters except "_".
Batch Description	Enter a description for the Batch based on the Batch Name. **NOTE:** The Batch description should be alphanumeric. The allowed special characters are_.:- and <blank space>, along with spaces and alpha-numeric. It should not exceed 200 characters in length.
Service URL Name/ Service URL	• Select the Service URL name from the drop-down list, if it is available. The Service URL is displayed in the **Service URL** field. • To add a new service URL, enter a name to identify it in the **Service URL Name** field and enter the proper URL in the **Service URL** field. You can give partial URL here and the remaining URL in the Task Service URL.

3. From the Batch Parameters pane, click  to add a new Batch Parameter. By default, $FICMISDATE$ and $BATCHRUNID$ are added as Batch parameters.

> **TIP** Enclose the parameter value for a run time with $ symbol. For example, $paramName$.

a. Enter the Parameter name in the **Param Name** field.

b. Enter the Parameter value in the **Param Value** field.

You can delete a parameter by clicking  corresponding to the parameter.

4. Click **Save**.

   The new Batch is created and displayed in the **Define Batch** window.

8.4.1.2 Edit a Batch

The Edit Batch option allows you to edit the Batch details such as Batch Description, Service URL Name and Service URL and also add a new Batch Parameter.

> **NOTE** Seeded batches cannot be edited.

To modify a Batch, perform the following steps:

1. In the **Define Batch** window, click  corresponding to the Batch you want to modify.

   The **Edit Batch** window is displayed.
2. Modify the required Batch details. For more information, see Create a Batch section.
3. Click **Save**.

   The edited batch is saved and displayed in the **Define Batch** window.

8.4.1.3 Copy a Batch

The Copy Batch option allows you to copy a Batch that you want to clone or create instances in the system from the **Define Batch** window.

To copy a Batch, perform the following steps:

1. In the **Define Batch** window, click  corresponding to the Batch that you want to copy.

   The **Copy Batch** window is displayed.
2. Specify the Batch details as you want to clone and copy the existing batch.

   For more information, see Create a Batch section.
3. Click **Save**.

   The copied batch is saved and displayed in the **Define Batch** window.

8.4.1.4 Delete a Batch

The **Delete Batch** option allows you to delete a Batch that are no longer required in the system from the **Define Batch** window.

> **NOTE** Seeded batches cannot be deleted.

To delete a Batch, perform the following steps:

1. From the **Define Batch** window, click  corresponding to the Batch you want to delete.
2. Click OK in the confirmation dialog to confirm deletion. If the batch has any active schedules a warning is displayed.

   Upon confirmation, all schedules of the batch are also deleted.

8.4.2 Batch Group

Batch Group is a process of grouping the batches that are required to be execute together for execution of the Date and time-based background tasks based on a defined period during which the resources were available for batch processing.

8.4.2.1 Create a Batch Group

You can create a new batch group in the Define Batch window and schedule and monitor the batch group that you created.

To create a new Batch Group, perform the following steps:

1. In the Define Batch window, click  .

   The **Create a New Batch** window is displayed.
2. Select Batch Group option.
3. Specify the following fields:
   - Name
   - Description
   - Add Batches

> **NOTE** The Add Batches is a multi-select field, you can select the batches that you want to add to the group using this field.

4. Click **Save**.

   The new Batch Group is created and displayed in the **Define Batch** window.

8.4.2.2 Edit a Batch Group

The **Edit Batch Group** option allows you to edit the Batch Group details such as Batch Group name, Added Batches, and Batch Group Description.

To modify a Batch Group, perform the following steps:

1. In the **Define Batch** window, click Batch Group option to list the batch groups.
2. Click  corresponding to the Batch Group you want to modify.

   The **Edit Batch** window is displayed.
3. Modify the required Batch Group details.

   For more information, see Create a Batch Group section.
4. Click **Save**.

   The edited batch group is saved and displayed in the **Define Batch** window.

8.4.2.3 Copy a Batch Group

The **Copy Batch** option allows you to copy a Batch that you want to clone or create instances in the system from the **Define Batch** window.

To copy a Batch Group, perform the following steps:

1. In the **Define Batch** window, click Batch Group option to list the batch groups.
2. Click  corresponding to the Batch Group that you want to copy.

   The **Copy Batch** window is displayed.
3. Specify the Batch Details as you want to clone and copy the existing batch.

   For more information, see Create_a_Batch_Group section.
4. Click **Save**.

   The copied batch group is saved and displayed in the **Define Batch** window.

8.4.2.4 Delete a Batch Group

The Delete Batch option allows you to delete a Batch that are no longer required in the system from the **Define Batch** window.

To delete a Batch Group, perform the following steps:

1. From the **Define Batch** window, click Batch Group option to list the batch groups.
2. Click  corresponding to the Batch Group you want to delete.
3. Click OK in the confirmation dialog to confirm deletion.

8.5 Define Tasks

The **Define Tasks** window displays the list of tasks associated with a specific Batch definition. You can create new tasks, edit the existing tasks or delete unwanted tasks. Additionally, you can specify task precedence for each task in Task Precedence window and click the Schedule  icon to Schedule the batch.

The **Define Tasks** window allows you to perform the following task operations for your batch and batch group:

- Batch
- Batch Group

8.5.1 Batch

Batch is a process of execution of the Date and time-based background tasks based on a defined period during which the resources were available for batch processing. You can perform the following operation for the batch based on the task.

- Add a Task
- Modify a Task
- Define Task Precedence
- Delete a Task

8.5.1.1 Add a Task

Adding a new task option allows you to add new tasks to a selected Batch definition.

To add new task, perform the following steps:

1. Click **Define Tasks** from the Header panel.

   The **Define Task** window is displayed.
2. Select the Batch for which you want to add new task from the **Select** drop-down list.
3. Click Add  .

   The **Create a New Task** window is displayed.
4. Enter the details as tabulated:

Table 3: Fields in the Create a New Task window and their Description

Field	Description
Task Details	
Task Name	Enter the task name. **NOTE:** • The Task Name must be alphanumeric and should not start with a number. • The Task Name should not exceed 60 characters in length. • The Task Name should not contain any special characters except underscore (_).
Task Description	• Enter the task description along with spaces and alpha-numeric. No special characters are allowed in Task Description. • Words like Select From or Delete From should not be entered in the Description.
Task Type	Select the task type from the drop-down list.
Batch Service URL	Select the required Batch Service URL from the drop-down list. This can be blank, and you can provide the full URL in the **Task Service URL** field.
Task Service URL	Enter task service URL if it is different from Batch Service URL.

5. From the *Task Parameters* pane, click Add to add a new Task Parameter. By default, all Batch level parameters are added and enabled as task parameters. To disable, deselect the check box corresponding to the task parameter ( )
   a. Enter the Parameter name in the **Param Name** field.
   b. Enter the Parameter value in the **Param Value** field.

   You can delete a parameter by clicking corresponding to the parameter.
6. Click **Save**.

8.5.1.2 Modify a Task

Modifying a task option allows you to modify the details of existing tasks of a Batch definition such as Task Description, Task Type, Batch Service URL and Task Service URL. You can also add a new task parameter and enable or disable already existing task parameters.

To modify a Task, perform the following steps:

1. From the **Define Task** window, select the Batch whose task details you want to modify, from the **Select** drop-down list.
2. Click Edit corresponding to the Task whose details you want to modify.

   The **Edit Task** window is displayed.
3. Modify the required Task Details.

   For more information, see Add a Task section.

4. Click **Save**.

8.5.1.3 Define Task Precedence

Task Precedence indicates the execution-flow of a Batch. Task Precedence value facilitates you to determine the order in which the specific Tasks of a Batch are executed.

For example, consider a Batch consisting of 4 Task. The first 3 Task does not have a precedence defined and hence will be executed simultaneously during the Batch execution. But, Task 4 has a precedence value as task 1 which indicates that, Task 4 is executed only after Task 1 has been successfully executed.

You can set Task precedence between Tasks or define to run a Task after a set of other tasks. However, multiple tasks can be executed simultaneously, and cyclical execution of tasks is not permitted. If the precedence for a Task is not set, the Task is executed immediately on Batch execution.

To define the task precedence in the *Define Task* window, perform the following steps:

1. Click Menu  button corresponding to the task for which you want to add precedence task.

   The **Task Precedence Mapping** window is displayed.

   > **NOTE** The Task Precedence option is disabled if a batch has only one task associated.

   a. Select the Task you want to execute before the current task, from the **Available Tasks** pane and click  . You can press Ctrl key for multiple selections.

   b. To select all the listed Tasks, click  .

   c. To remove a Task, select the task from the *Selected Tasks* pane and click  .

   d. To remove all the selected Tasks, click  .

2. Click **Save** to update Task Precedence.
3. Click **Preview** to view the Precedence information.

8.5.1.4 Delete a Task

You can remove a task from a Batch definition which are no longer required in the system by deleting it from the **Define Task** window.

To delete a Task, perform the following steps:

1. From the **Define Task** window, select the Batch whose task details you want to delete from the **Select** drop-down list.
2. Click Delete  corresponding to the Task you want to delete.

3. Click **OK** in the confirmation dialog to confirm deletion.

8.5.2 Batch Group

Batch Group is a process of grouping the batches that are required to be execute together for execution of the Date and time-based background tasks based on a defined period during which the resources were available for batch processing. You can perform the following operation for the batch based on the task.

- Define Task Precedence

8.5.2.1 Define Task Precedence

Task Precedence indicates the execution-flow of a Batch. Task Precedence value facilitates you to determine the order in which the specific Tasks of a Batch are executed.

For example, consider a Batch consisting of 4 Task. The first 3 Task does not have a precedence defined and hence will be executed simultaneously during the Batch execution. But, Task 4 has a precedence value as task 1 which indicates that, Task 4 is executed only after Task 1 has been successfully executed.

You can set Task precedence between Tasks or define to run a Task after a set of other tasks. However, multiple tasks can be executed simultaneously, and cyclical execution of tasks is not permitted. If the precedence for a Task is not set, the Task is executed immediately on Batch execution.

To define the task precedence in the *Define Task* window, perform the following steps:

1. Click Menu  button corresponding to the task for which you want to add precedence task.

   The **Task Precedence Mapping** window is displayed.

   > **NOTE** The Task Precedence option is disabled if a batch has only one task associated.

   a. Select the batch that you want to execute before the current task, from the **Available Tasks** pane and click  . You can press Ctrl key for multiple selections.

   b. To select all the listed batches, click  .

   c. To remove a batch, select the task from the **Selected Tasks** pane and click  .

   d. To remove all the selected batches, click  .

2. Click **Save** to update Task Precedence in the batches.
3. Click **Preview** to view the Precedence information.

8.6 Schedule Batch

The *Schedule Batch* window facilitates you to run, schedule, re-start, re-run the batches in the Scheduler Service. After you upload the data in the required format into the Object Storage, you must load the data into the system using the Scheduler Service. You can schedule them to run in a required pattern and view the run time status of the scheduled services using the Monitor Batch feature.

The **Schedule Batch** window allows you to perform the following operations for your batch and batch group:

- Batch
- Batch Group

8.6.1 Batch

Batch is a process of execution of the Date and time-based background tasks based on a defined period during which the resources were available for batch processing. You can perform the following operation for the batch:

- Execute a Batch
- Schedule a Batch
- Re-start a Batch
- Re-run a Batch
- Edit Dynamic Parameters
- Task Definition of a Batch

8.6.1.1 Execute a Batch

The Execute batch option allows you to run a batch instantaneously. To execute a batch, perform the following steps:

1. Click **Schedule Batch** from the Header panel.

   The **Schedule** window is displayed.

2. Select the Batch Name from the Select Name drop down menu.

   For example, AMLDataLoad.

3. Click **Execute**.

   The Execution Status Dialog Box is displayed with the Batch executed successfully message.

   This indicates the unique identification reference number for the batch and date of the batch execution.

4. In the Execution Status Dialog Box, click Monitor to monitor the batch.
5. If you want to exclude/include some tasks, click **Exclude Tasks**. For more information, see Exclude/Include Tasks section.
6. If you want to hold/release some tasks, click **Hold Tasks**.

   For more information, see Hold/Release Tasks section.

7. If you want to edit the dynamic parameters of the batch, click **Edit Dynamic Parameters** For more information, see Edit Dynamic Parameters section.

8.6.1.2 Schedule a Batch

You can schedule a Batch to run just for Once, Daily, Weekly, Monthly, or Cron Expression for scheduling the batches. You can also have a user defined schedule to schedule and run a batch.

8.6.1.2.1 Schedule Once

To schedule a Batch to run once, perform the following steps:

1. Click **Schedule Batch** from the Header panel.

   The **Schedule Batch** window is displayed.
2. In the **Schedule Batch** window, click **Once**.
3. Select the Batch or Batch Name you want to schedule for once from the Select drop down menu.
4. Enter a Schedule Name.
5. Click  and select the date on which you want to run the Batch.
6. Click  and select the time at which you want to run the Batch.
7. Click **Schedule**.

8.6.1.2.2 Schedule Daily

To schedule a Batch to run daily, perform the following steps:

1. In the **Schedule Batch** window, click **Daily**.
2. Select the Batch or Batch Name you want to schedule daily from the Select drop down menu.
3. Enter a **Schedule Name**.
4. Click  and select the start date from which you want to run the Batch.
5. Click  and select the end date till which you want to run the Batch.
6. Click  and select the time at which you want to run the Batch daily.
7. Click **Schedule**.

8.6.1.2.3 Schedule Weekly

To schedule a Batch to run weekly, perform the following steps:

1. In the **Schedule Batch** window, click **Weekly**.
2. Select the Batch or Batch Name you want to schedule weekly from the Select drop down menu.
3. Enter a Schedule Name.
4. Click  and select the start date from which you want to run the Batch.

5. Click  and select the end date till which you want to run the Batch.
6. Click  and select the time at which you want to run the Batch.
7. Select the days on a week you want to run the Batch from the Select Days of the Week multi-select drop down menu.
8. Click **Schedule**.

8.6.1.2.4 Schedule Monthly

To schedule a Batch to run monthly, perform the following steps:

1. In the **Schedule Batch** window, click **Monthly**.
2. Select the Batch or Batch Name you want to schedule monthly from the Select drop down menu.
3. Enter a Schedule Name.
4. Click  and select the start date from which you want to run the Batch.
5. Click  and select the end date till which you want to run the Batch.
6. Click  and select the time at which you want to run the Batch.
7. Select the months in a year you want to run the Batch from the Select Months of the Year multi-select drop down menu.
8. Select the day of the month that you want to run the Batch from the Select Day of the Month drop down menu.
9. Click **Schedule**.

8.6.1.2.5 Schedule Cron Expression

To run a Batch in a user-defined schedule, you can have custom schedule with the help of Cron Expression. A cron expression is a string comprised of 6 or 7 fields separated by white space. Fields can contain any of the allowed values, along with various combinations of the allowed special characters for that field. For more information, click  icon next to the Cron Expression field.

To schedule a Batch based on Cron Expression, perform the following steps

1. In the **Schedule Batch** window, click **Cron Expression.**
2. Select the Batch or Batch Name you want to schedule from the Select drop down menu.
3. Enter a **Schedule Name**.
4. Enter the Cron Expression for your schedule. For more information about the Cron Expression, click  icon next to the Cron Expression field.
5. Click **Schedule**.

8.6.1.3 Re-start a Batch

You can restart a Batch which has not been executed successfully or which has been explicitly interrupted, or cancelled, or put on hold during the execution process. By restarting a Batch, you can continue Batch execution directly from the point of interruption or failure and complete executing the remaining tasks.

To re-start a batch, perform the following steps:

1. Click **Schedule Batch** from the Header panel.

   The Schedule window is displayed.
2. From the **Schedule** window, select **Re-start** tab.
3. Select the Batch or Batch Name you want to re-start from the Select Name drop down menu.
4. Select the Batch Run ID.
5. Click **Re-start**.

8.6.1.4 Re-run a Batch

You can re-run a Batch which has previously been executed. Rerun Batch facilitates you to run the Batch irrespective of the previous execution state. A new Batch Run ID is generated during the Rerun process and the Batch is executed as similar to the new Batch Run.

To re-run a batch, perform the following steps:

1. Click **Schedule Batch** from the Header panel.

   The **Schedule Batch** window is displayed.
2. In the **Schedule Batch** window, select **Re-run** tab.
3. Select the Batch or Batch Name you want to re-run from the Select Name drop down menu.
4. Select the Batch Run ID.
5. Click **Re-run**.

8.6.1.5 Edit Dynamic Parameters

Dynamic Parameters facilitates you to the modify the dynamic parameters for the batch. You can change the param value from the Edit Dynamic Params window and save the changes to the Batch. The Edit Dynamic Parameters option is available in all the tab in the *Schedule Batch* window.

To edit the dynamic parameters for a batch, perform the following steps:

1. In the **Schedule Batch** window, click Edit Dynamic Parameters.

   The **Edit Dynamic Params** window is displayed.
2. In the **Edit Dynamic Params** window, modify the values as required.
3. Click **Save**.

   The modified parameters are applied to the Batch.

8.6.1.6 Task Definitions of a Batch

You can modify the task definition state in the Batch Execution window to exclude or hold the defined task in a Batch from execution. The excluded tasks are therefore assumed to have completed execution and get excluded during the Batch Run.

While executing or scheduling a Batch from the **Schedule Batch** window, you can:

- Exclude a task or include the excluded task
- Hold a task or release the held task

8.6.1.6.1 Exclude or Include Tasks

You can exclude tasks or include the excluded tasks during Batch Execution. The excluded task components are therefore executed in the normal process assuming that the excluded task have completed execution.

To exclude/include tasks, perform the following steps:

1. In the **Schedule Batch** window, click **Exclude Tasks**.

   The **Select Tasks** window is displayed.

2. To exclude tasks:

   a. Select the required task from the **Included Tasks** list and click  . You can press Ctrl key for multiple selections.

   b. To exclude all tasks, click  .

3. To include the excluded tasks:

   a. Select the required task from the **Excluded Tasks** list and click  . You can press Ctrl key for multiple selections.

   b. To include all excluded tasks, click  .

4. Click **Save**.

8.6.1.6.2 Hold or Release Tasks

You can hold tasks or release the held tasks during Batch Execution. The tasks which are on hold along with the defined components are skipped during execution. However, at least one task should be available in a Batch without being held/excluded for Batch execution.

To hold/release tasks, perform the following steps:

1. In the **Schedule Batch** window, click **Hold Tasks**.

   The **Select Tasks** window is displayed.

2. To hold tasks:

a. Select the required task from the **Released Tasks** list and click  . You can press Ctrl key for multiple selections.

b. To hold all tasks, click  .

3. To release held tasks:

a. Select the required task from the **Held Tasks** list and click  . You can press Ctrl key for multiple selections.

b. To release all held tasks, click  .

4. Click **Save**.

8.6.2 Batch Group

Batch Group is a process of grouping the batches that are required to be execute together for execution of the Date and time-based background tasks based on a defined period during which the resources were available for batch processing. You can perform the following operation for the batch group:

- Execute a Batch Group
- Schedule a Batch Group
- Re-start a Batch Group
- Re-run a Batch Group
- Edit Dynamic Parameters
- Task Definition of a Batch Group

8.6.2.1 Execute a Batch Group

The Execute batch option allows you to run a batch instantaneously. To execute a batch group, perform the following steps:

1. Click **Schedule Batch** from the Header panel.

   The **Schedule** window is displayed.

2. Select the Batch Group from the Select Name drop down menu.

   For example, AMLDataLoad.

3. Click **Execute**.

   The Execution Status Dialog Box is displayed with the Batch executed successfully message.

   This indicates the unique identification reference number for the batch and date of the batch execution.

4. In the Execution Status Dialog Box, click Monitor to monitor the batch.

5. If you want to exclude/include some tasks, click **Exclude Tasks**.

For more information, see Exclude/Include Tasks section.

6. If you want to hold/release some tasks, click **Hold Tasks**.

   For more information, see Hold/Release Tasks section.

7. If you want to edit the dynamic parameters of the batch, click **Edit Dynamic Parameters**.

   For more information, see Edit Dynamic Parameters

8.6.2.2 Schedule a Batch Group

You can schedule a Batch group to run just for Once, Daily, Weekly, Monthly, or Cron Expression for scheduling the batches. You can also have a user defined schedule to schedule and run a batch.

8.6.2.2.1 Schedule Once

To schedule a Batch to run once, perform the following steps:

1. Click **Schedule Batch** from the Header panel.

   The **Schedule Batch** window is displayed.

2. In the **Schedule Batch** window, click **Once**.
3. Select the Batch Group you want to schedule for once from the Select drop down menu.
4. Enter a Schedule Name.
5. Click  and select the date on which you want to run the Batch.
6. Click  and select the time at which you want to run the Batch.
7. Click **Schedule**.

8.6.2.2.2 Schedule Daily

To schedule a Batch to run daily, perform the following steps:

1. In the **Schedule Batch** window, click **Daily**.
2. Select the Batch Group you want to schedule daily from the Select drop down menu.
3. Enter a **Schedule Name**.
4. Click  and select the start date from which you want to run the batch group.
5. Click  and select the end date till which you want to run the batch group.
6. Click  and select the time at which you want to run the batch group daily.
7. Click **Schedule**.

8.6.2.2.3 Schedule Weekly

To schedule a Batch to run weekly, perform the following steps:

1. In the **Schedule Batch** window, click **Weekly**.

2. Select the Batch Group you want to schedule weekly from the Select drop down menu.
3. Enter a Schedule Name.
4. Click  and select the start date from which you want to run the batch group.
5. Click  and select the end date till which you want to run the batch group.
6. Click  and select the time at which you want to run the batch group.
7. Select the days in a week you want to run the batch group from the Select Days of the Week multi-select drop down menu.
8. Click **Schedule**.

8.6.2.2.4 Schedule Monthly

To schedule a Batch to run monthly, perform the following steps:

1. In the **Schedule Batch** window, click **Monthly**.
2. Select the Batch Group you want to schedule monthly from the Select drop down menu.
3. Enter a Schedule Name.
4. Click  and select the start date from which you want to run the batch group.
5. Click  and select the end date till which you want to run the batch group.
6. Click  and select the time at which you want to run the batch group.
7. Select the months in a year you want to run the batch group from the Select Months of the Year multi-select drop down menu.
8. Select the day of the month that you want to run the batch group from the Select Day of the Month drop down menu.
9. Click **Schedule**.

8.6.2.2.5 Schedule Cron Expression

To run a Batch in a user-defined schedule, you can have custom schedule with the help of Cron Expression. A cron expression is a string comprised of 6 or 7 fields separated by white space. Fields can contain any of the allowed values, along with various combinations of the allowed special characters for that field. For more information, click  icon next to the Cron Expression field.

To schedule a Batch based on Cron Expression, perform the following steps

1. In the **Schedule Batch** window, click **Cron Expression.**
2. Select the Batch Group you want to schedule from the Select drop down menu.
3. Enter a **Schedule Name**.
4. Enter the Cron Expression for your schedule.

For more information about the Cron Expression, click  icon next to the Cron Expression field.

5. Click **Schedule**.

8.6.2.2.6 Pre Conditions for a Batch Group

You can schedule the batches and set the pre conditions within a Batch group with frequency as Weekly, Monthly, or based on an Interval for scheduling the batches. The batch that satisfies the configured pre conditions are executed as part of the schedule.

> **NOTE** Pre Conditions can only be applied when using the Schedule option in the Schedule Batch window.

Weekly

To set the pre conditions to the batches in a batch group weekly, perform the following steps:

1. In the **Schedule Batch** window, you can select either Once, Daily, Weekly, Monthly, or Cron Expression option based on the schedule that you want to run the batch group.
2. Select the Batch Group you want from the Select drop down menu.
3. Enter a Schedule Name.
4. Specify the other details displayed when you are selecting Once, Daily, Weekly, Monthly, or Cron Expression.
5. Click Pre Conditions.
6. In the Pre Conditions window, specify the Batch from the drop down and from the Frequency drop down and select Weekly.
7. Select the days from the Select Days drop down that you want to schedule the batch run within the selected week.
8. Click  to add the specified entry.

> **NOTE** Pre Conditions can be added only to one batch at a time

9. Click **Save**.

   The batch is executed based on the configured pre conditions.

Monthly

To set the pre conditions to the batches in a batch group monthly, perform the following steps:

1. In the **Schedule Batch** window, you can select either Once, Daily, Weekly, Monthly, or Cron Expression option based on the schedule that you want to run the batch group.
2. Select the Batch Group you want from the Select drop down menu.
3. Enter a Schedule Name.

4. Specify the other details displayed when you are selecting Once, Daily, Weekly, Monthly, or Cron Expression.
5. Click Pre Conditions.
6. In the Pre Conditions window, specify the Batch from the drop down and from the Frequency drop down and select Monthly.
7. Select the days from the Select Days drop down that you want to schedule the batch run within the selected week.
8. Click  to add the specified entry.

> **NOTE** Pre Conditions can be added only to one batch at a time.

9. Click **Save**.

   The batch is executed based on the configured pre conditions.

Interval

To set the pre conditions to the batches in a batch group based on an interval, perform the following steps:

1. In the **Schedule Batch** window, you can select either Once, Daily, Weekly, Monthly, or Cron Expression option based on the schedule that you want to run the batch group.
2. Select the Batch Group you want from the Select drop down menu.
3. Enter a Schedule Name.
4. Specify the other details displayed when you are selecting Once, Daily, Weekly, Monthly, or Cron Expression.
5. Click Pre Conditions.
6. In the Pre Conditions window, specify the Batch from the drop down and from the Frequency drop down and select Interval.
7. Select the interval from the Custom Recurrence (Repeat every) Days drop down that you want to schedule the batch run within the selected week.

> **NOTE** The Custom Recurrence can be set maximum to 60 days.

8. Click  to add the specified entry.

> **NOTE** Pre Conditions can be added only to one batch at a time.

9. Click Save.

10. The batch is executed based on the configured pre conditions.

8.6.3 Re-start a Batch Group

You can restart a Batch which has not been executed successfully or which has been explicitly interrupted, or cancelled, or put on hold during the execution process. By restarting a Batch, you can continue Batch execution directly from the point of interruption or failure and complete executing the remaining tasks.

To re-start a batch, perform the following steps:

1. Click **Schedule Batch** from the Header panel.

   The **Schedule** window is displayed.
2. From the **Schedule** window, select **Re-start** tab.
3. Select the Batch Group you want to re-start from the Select Name drop down menu.
4. Select the Batch Run ID.
5. Click **Re-start**.

8.6.4 Re-run a Batch Group

You can re-run a Batch which has previously been executed. Rerun Batch facilitates you to run the Batch irrespective of the previous execution state. A new Batch Run ID is generated during the Rerun process and the Batch is executed as similar to the new Batch Run.

To re-run a batch, perform the following steps:

1. Click **Schedule Batch** from the Header panel.

   The **Schedule Batch** window is displayed.
2. In the **Schedule Batch** window, select **Re-run** tab.
3. Select the Batch Group you want to re-run from the Select Name drop down menu.
4. Select the Batch Run ID.
5. Click **Re-run**.

8.6.5 Edit Dynamic Parameters

Dynamic Parameters facilitates you to the modify the dynamic parameters for the batch. You can change the param value from the Edit Dynamic Params window and save the changes to the Batch. The Edit Dynamic Parameters option is available in all the tab in the **Schedule Batch** window.

To edit the dynamic parameters for a batch group, perform the following steps:

1. In the **Schedule Batch** window, click **Edit Dynamic Parameters**.

   The **Edit Dynamic Params** window is displayed.
2. In the **Edit Dynamic Params** window, modify the values as required.
3. Click **Save**.

   The modified parameters are applied to the Batch.

8.6.6 Task Definitions of a Batch Group

You can modify the task definition state in the Batch Execution window to exclude or hold the defined task in a Batch from execution. The excluded tasks are therefore assumed to have completed execution and get excluded during the Batch Run.

While executing or scheduling a batch group from the *Schedule Batch* window, you can:

- Exclude a task or include the excluded task
- Hold a task or release the held task.

8.6.6.1 Exclude or Include Tasks

You can exclude tasks or include the excluded tasks during Batch Group Execution. The excluded task components are therefore executed in the normal process assuming that the excluded task have completed execution.

To exclude/include tasks, perform the following steps:

1. In the Schedule Batch window, click **Exclude Tasks**.

   The **Select Tasks** window is displayed.

2. To exclude tasks:

   a. Select the required task from the **Included Tasks** list and click  . You can press Ctrl key for multiple selections.

   b. To exclude all tasks, click  .

3. To include the excluded tasks:

   a. Select the required task from the **Excluded Tasks** list and click  . You can press Ctrl key for multiple selections.

   b. To include all excluded tasks, click  .

4. Click **Save**.

8.6.6.2 Hold or Release Tasks

You can hold tasks or release the held tasks during Batch Group Execution. The tasks which are on hold along with the defined components are skipped during execution. However, at least one task should be available in a Batch without being held or excluded for Batch execution.

To hold/release tasks, perform the following steps:

1. In the **Schedule Batch** window, click **Hold Tasks**.

   The **Select Tasks** window is displayed.

2. To hold tasks:

a. Select the required task from the *Released Tasks* list and click  . You can press Ctrl key for multiple selections.

b. To hold all tasks, click  .

3. To release held tasks:

a. Select the required task from the *Held Tasks* list and click  . You can press Ctrl key for multiple selections.

b. To release all held tasks, click  .

4. Click **Save**.

8.7 Monitor Batch

The Monitor Batch enables you to view the status of executed Batch along with the task's details. You can track the issues if any, on regular intervals and ensure smoother Batch execution. A visual representation as well as tabular view of the status of each Tasks in the Batch is available.

The **Monitor Batch** window allows you to perform monitoring for the following operations:

- Batch
- Batch Group

8.7.1 Batch

Batch is a process of execution of the Date and time-based background tasks based on a defined period during which the resources were available for batch processing. To monitor a batch, perform the following steps:

1. Click **Monitor Batch** from the Header panel.

   The **Monitor** window is displayed.

2. Select the **Batch from the Select drop-down and then select the Batch Run ID** from the Run ID drop-down.

3. Click Start Monitor.

   The result is displayed in Visualization and List View tabs. Details of these tabs are as follows:

   - The Visualization tab displays the details in the form of a chart represented with the following details:

     i. Batch Status: Displays the batch status, the different batch status are NOT-STARTED, ON-GOING, SUCCESSFUL, FAILED, INTERRUPTED, EXCLUDED, HELD, and UNDEFINED.

     ii. Batch Start Time: Displays the batch start time details.

     iii. Batch End Time: Displays the batch end time details.

iv. Task Details: Mouseover the task to display its status and details.

- The **List View** tab displays the details in a tabular form with the following details:

i. Batch Status: Displays the batch status, the different batch status are NOT-STARTED, ON-GOING, SUCCESSFUL, FAILED, INTERRUPTED, EXCLUDED, HELD, and UNDEFINED.

ii. Batch Start Time: Displays the batch start time details.

iii. Batch End Time: Displays the batch end time details.

iv. Task Details: Mouseover the task to display its status and details.

v. More Information: The message returned by the Rest Service.

4. If you wish to stop the monitoring, select Stop Monitor. You can also specify the Start and Stop Monitor options along with refresh interval in the Refresh every seconds and minutes fields.

NOTE

- You can select the refresh interval and the duration for the auto refresh. The refresh interval is defaulted to 5 seconds and duration is defaulted to 5 minutes. That is the refresh happens every 5 seconds for next 5 minutes.
- Range of interval input must be between 5 to 60 seconds and range of duration.
- Input should be between 5 to 180 minutes.
- You can use the Stop Monitor button to stop the auto refresh.

5. To restart the Batch Group, select Restart  .
6. To rerun the Batch Group, select Rerun  .
7. To interrupt the Batch Group, select Interrupt  .
8. To view the log information about the batch, click the Log  icon in the List View tab.

   The Log Viewer pop-up is displayed.
9. In the Log Viewer pop-up, the log information is displayed. You can click Download  icon to download the log or click the Close  icon to close the log information.

8.7.2 Batch Group

Batch Group is a process of grouping the batches that are required to be execute together for execution of the Date and time-based background tasks based on a defined period during which the resources were available for batch processing. To monitor a batch group, perform the following steps:

1. Click **Monitor Batch** from the Header panel.

The **Monitor** window is displayed.

2. Select the **Batch Group from the Select drop-down and then select the Batch Run ID** from the Run ID drop-down.
3. Click Start Monitor.

   The result is displayed in Visualization and List View tabs. Details of these tabs are as follows:

   - The Visualization tab displays the details in the form of a chart represented with the following details:
     i. Batch Status: Displays the batch status, the different batch status are NOT-STARTED, ON-GOING, SUCCESSFUL, FAILED, INTERRUPTED, EXCLUDED, HELD, and UNDEFINED.
     ii. Batch Start Time: Displays the batch start time details.
     iii. Batch End Time: Displays the batch end time details.
     iv. Batch Details: Mouseover the task to display its status and details.
   - The **List View** tab displays the details in a tabular form with the following details:
     i. Batch Status: Displays the batch status, the different batch status are NOT-STARTED, ON-GOING, SUCCESSFUL, FAILED, INTERRUPTED, EXCLUDED, HELD, and UNDEFINED.
     ii. Batch Start Time: Displays the batch start time details.
     iii. Batch End Time: Displays the batch end time details.
     iv. Batch Details: Mouseover the task to display its status and details.
     v. More Information: The message returned by the Rest Service.
4. If you wish to stop the monitoring, select Stop Monitor. You can also specify the Start and Stop Monitor options along with refresh interval in the Refresh every seconds and minutes fields.

> **NOTE**
>
> - You can select the refresh interval and the duration for the auto refresh. The refresh interval is defaulted to 5 seconds and duration is defaulted to 5 minutes. That is the refresh happens every 5 seconds for next 5 minutes.
> - Range of interval input must be between 5 to 60 seconds and range of duration.
> - Input should be between 5 to 180 minutes.
> - You can use the Stop Monitor button to stop the auto refresh.

5. To restart the Batch Group, select Restart  .
6. To rerun the Batch Group, select Rerun  .

7. To interrupt the Batch Group, select Interrupt  .

To view the log information about the batch group, click View Log in the List View tab

9 Audit Trail

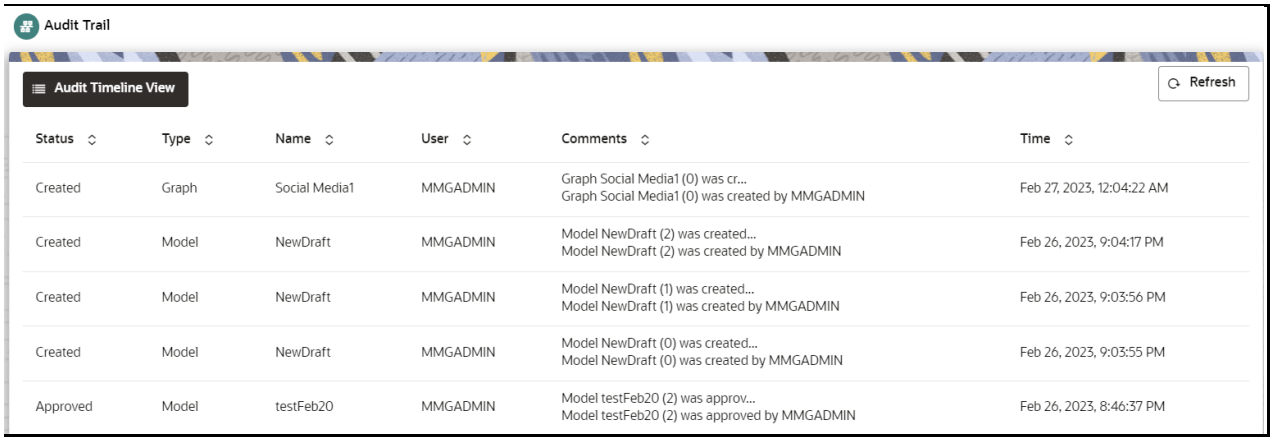
At any time, you can audit the models from the Audit Trail window. The Audit Trail window provides the complete details of model. This shows the information such as, when Model was created, who created the Model, workflow of Model, for example when this Model became champion or deployed, and so on.

The sequence of actions performed in the model lifecycle is listed in the table view and the timeline view (graphical representation).

To audit models, follow these steps:

1. Click **Launch Workspace** next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click **Audit Trail** to view the time of the various actions performed on the model in the **Audit Trail** window.

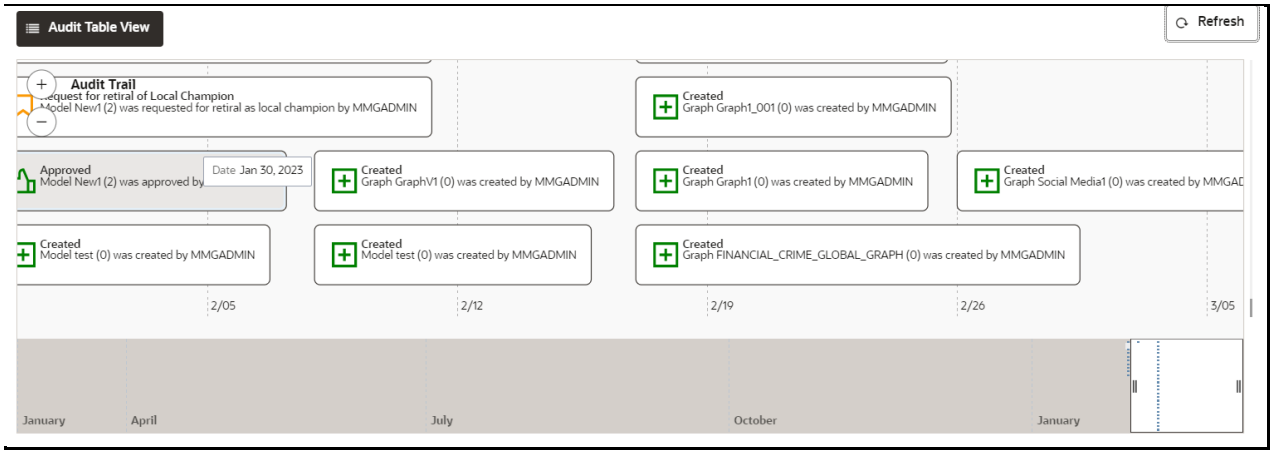
Figure 52: Audit Trail Window



3. Click **Audit Timeline View** for Timeline View. Click **Audit Table View** to switch to the regular view.

Click **Refresh** to refresh the Audit Trail window.

Figure 53: Audit Trail Window Timeline View with Horizontal Time Axis



10 Data Studio Options

MMG contains an underlying Notebook Server which has the following configurable options:

- Interpreters
- Tasks
- Permissions
- Credentials
- Templates

To access the Data Studio Options page, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Hover the mouse over the Data Studio Options widget  the following options are available:
   - Interpreters
   - Tasks
   - Permissions
   - Credentials
   - Templates

10.1 Configuring Interpreters

An interpreter is a program that directly executes instructions written in a programming or scripting language without requiring them previously to be compiled into a machine language program. Interpreters are plug-ins that enable users to use a specific language to process data in the backend. Examples of Interpreters are jdbc-interpreter, spark-interpreters, python-interpreters, etc. Interpreters allow you to define customized drivers, URLs, passwords, connections, SQL results to display, etc.

In OFS Compliance Studio, Interpreters are used in Notebooks to execute code in different languages. Each Interpreter has a set of properties that are adjusted and applied across all notebooks. For example, using the python-interpreter makes it possible to change between versions, whereas the jdbc-interpreter offers to customize the URL, schema, or credentials. In MMG, you can either use a default interpreter variant or create a new variant for an interpreter. You can create more than one variant for an interpreter. The benefit of creating multiple variants for an Interpreter is to connect different versions of interpreters (Python ver:3, Python ver:2, etc.). This helps to connect a different set of users and database schema. For example, Compliance Studio schema, BD schema, etc. Compliance Studio provides secure and safe credential management such as Oracle Wallet (jdbc wallet), Password (jdbc password), or KeyStores to link to interpreter variants to access secured data.

MMG has ready-to-use interpreters such as fcc-python, jdbc Interpreter, etc. You can configure them based on the use case. Additional variants of interpreters are created as multiple users might require different settings to access the database securely. The jdbc Interpreters use the credentials to enable secure data access.

> **NOTE** fcc-python, Pyspark, spark, and ore are a few other available interpreters.

Interpreters are configured when you want to modify URL, data location, drivers, enable or disable connections, etc.

1. To configure ready-to-use interpreters, follow these steps: 1. Click the Interpreter that you want to view from the list displayed on the LHS. The default configured interpreter variant is displayed on the RHS.
2. Modify the values in the fields as per requirement. For example, to modify a parameter's limit, connect to a different schema, PGX server, etc.
3. Click Update. The modified values are updated in the Interpreter.

For more information on Interpreters, see the Interpreter Configuration and Connectivity section in the OFS Compliance Studio Administration and Configuration Guide.

10.2 Managing Tasks

Tasks are created when notebooks or paragraphs are executed by the Notebook users. It is important to know the status of the execution, whether the tasks are created, rejected, canceled, etc. The Tasks page allows you to view the status of the task and associated notebooks, paragraphs, interpreters, etc. By default, all the tasks are listed on the Task page. You can view the specific task using filters such as task status, date of creation, and notebook name.

For more information on tasks, see the Monitoring Tasks on Notebook Server section in the OFS Compliance Studio Administration and Configuration Guide.

10.3 Managing Permissions

Use this section to view the logged-in users and view, add, or modify ready-to-use permissions granted to the users, roles, or groups. You can create groups, roles, and permission templates (actions).

> **NOTE** You can only view users and their details.

See the User Access and Permissioning Management section in the OFS Compliance Studio Administration and Configuration Guide.

10.4 Managing Credentials

Compliance Studio provides secure and safe credential management. Examples of credentials are passwords, Oracle Wallets, or KeyStores. This section links credentials (a wallet and a password) to the jdbc interpreter variant to enable secure data access. This linking enables the jdbc interpreter to securely connect to the specified Oracle Database. You can also create new credentials based on your requirement to connect to the new interpreter variants.

For more information on Credentials, see the Link Credentials section in the OFS Compliance Studio Administration and Configuration Guide.

> **NOTE** You can link credentials only to the jdbc interpreter. The Credentials section is enabled if an Interpreter variant can accept credentials.

10.5 Configuring Templates

Perform the following steps to Create, Import, and set the FCGM Default Template in the Template Dashboard.

1. Click CS Launch Workspace to display the CS Production Workspace window.
2. Click Templates from Design Studio Options.

Figure: Templates

The Compliance Studio offers different formats to view the result after a paragraph's execution. Templates enable you to define parameters to customize the result formats. You can customize the visualization of the result by defining parameters in a template and then applying that template to a notebook.

> **NOTE**
> - Compliance Studio comes with a default template, but users can customize this at the template level but can also override any global template settings in each notebook paragraph
> - It is recommended to use the template that is available from out-of-the-box Compliance Studio.

For more information, see the Interpreter Configuration and Connectivity section in the OFS Compliance Studio Administration and Configuration Guide.

11 Object Migration

Object Migration is the process of migrating or moving objects between environments.

You may want to migrate objects for several reasons such as managing global deployments on multiple environments or creating multiple environments so that you can separate the development, testing, and production processes.

11.1 Prerequisites to Migrate Objects

In order to migrate the objects, the users must be mapped to the **Object Migration Admin Group**.

> **NOTE** The Identity Administrator Group Users cannot migrate objects if they are not mapped to the **Object Migration Admin Group**.

11.2 Migration Object Types

You can migrate (import/export) the following Object Types by clicking the Object Migration  icon on the left menu.

- Schedule
- Batch
- Batch Group
- Pipeline
- Threshold
- Job
- PMF_Process
- Roles
- Groups

11.2.1 Schedule

Schedule provides the instruction to schedule the execution of defined processes. When a schedule is migrated, the associated batch is also migrated.

11.2.2 Batch

A batch is a group of jobs that are scheduled to automatically execute at a preset interval of time, without any user's intervention. When a batch is migrated, the batch and the associated pipeline information are also migrated.

11.2.3 Batch Group

A set of individual batches are consolidated to form a single Batch Group. When we migrate a Batch Group all the batches, tasks and pipeline information associated with that Batch Group are also migrated.

11.2.4 Pipeline

A pipeline is an embedded data processing engine that runs inside the application to filter, transform, and migrate data on-the-fly. Pipelines are a set of data processing elements called widgets connected in series, where the output of one widget is the input to the next element.

11.2.5 Threshold

The threshold limit associated with set variables values for scenarios in FCCM Cloud Service. These threshold values are set when scenarios are created or installed and can be changed, if required.

11.2.6 Job

Jobs provide set of instructions to execute Workflow Pipelines, based on the set threshold values

11.2.7 PMF_Process

PMF_Processes are defined to sequence the Workflow Pipelines the applications, and to design the artifacts that participate in the Pipelines, to implement the Pipelines. Export of PMF_Process will take care of dependent metadata, such as data fields, transition rules associated to the PMF process, that are defined in PMF.

11.2.8 Roles

Roles are used to map functions to a defined set of groups to ensure user access system security.

11.2.9 Groups

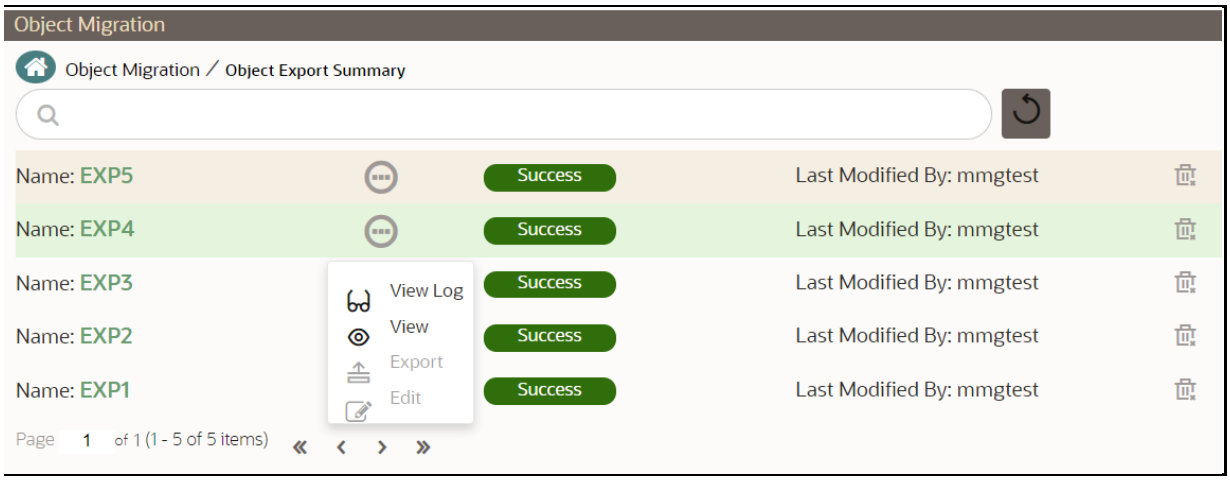
Groups are used to map Roles. Specific User Groups can perform only set of functions associated with that group.

11.3 Object Export Summary

To access the Object Export Summary page, follow these steps:

1. Click **Launch Workspace**  next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click the Object Migration  icon and select **Export Objects**.

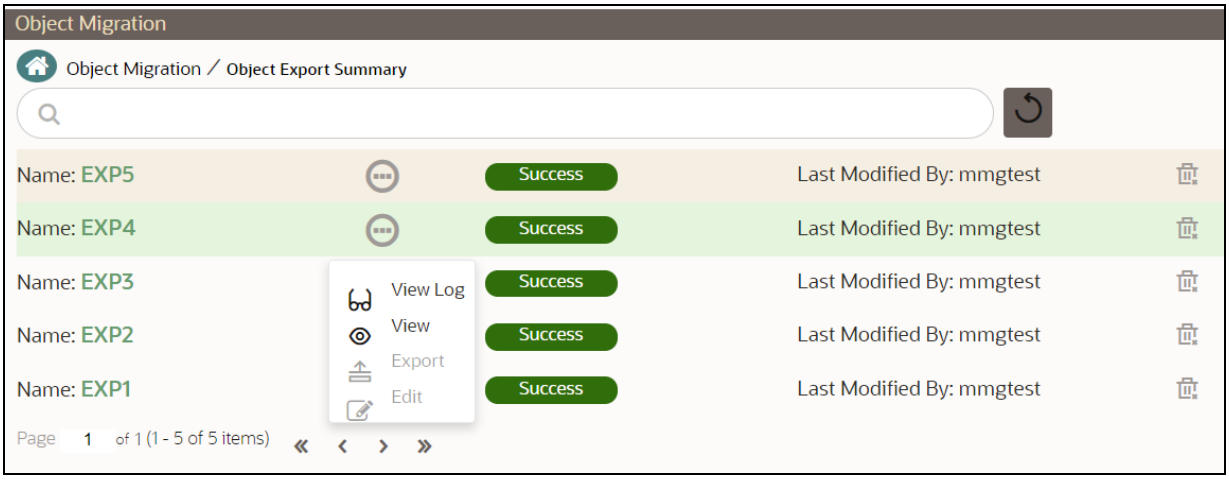
   The Object Export Summary page containing the records is displayed with the following details.



- **Name:** The unique migration name assigned to the collection when the migration definition was created.
- **Object Migration Status:** The migration status of the record corresponding to the specified Definition Name. The three migration status values are as follows:
  - **Success** - Set to Success, when the object export is completed successfully.
  - **Saved** - Set to Saved, when the migration definition is ready for export and needs to import.
  - **Failed** - Set to failed, when the migration definition is not exported successfully.
- **Last Modified By**: The ID of the Last Modified by user who has modified the record.

11.3.1 Navigating Migration Object Export Summary Page

To search for a specific migration definition, type the first few letters of the user name that you want to search in the **Search** box and click Search icon. The search results display the names that consist of your search string in the list of available users.



At the bottom of the page, you can enter the number of entries that are available on a single page in the Records box. You can increase or decrease the number of entries that are displayed using the up and down arrows. To navigate between pages in the **View** bar, use the following buttons:

- Use the **First Page**  button to view the entries in the first page.
- Use the **Previous Page**  button to view the entries in the previous page.
- Use the **Next Page**  button to view the entries in the next page.
- Use the **Last Page**  button to view the entries in the last page.

You can also navigate to the desired page. To do this, enter the page number in the **View** bar control and press **Enter**.

To create a new migration definition, click **Add**  button and proceed with Creating Migration Export Object Definitions

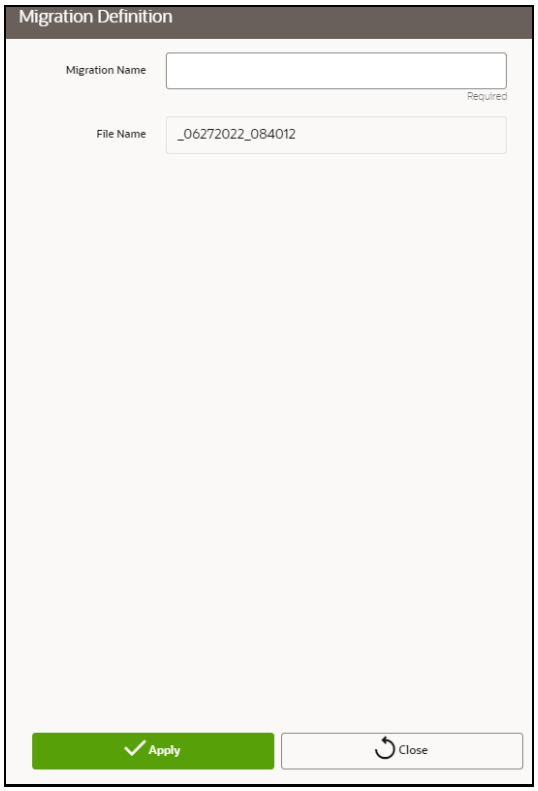
11.3.2 Creating Migration Export Object Definitions

You can create Migration Export object definitions for the following object types.

- Batch Group
- Threshold
- PMF Process
- Batch
- Pipeline
- Schedule
- Job

To create a definition for export of objects to be migrated, do as follows:

1. Click **Add**  in the **Object Export Summary** Page to display the **Migration Definition Window**.



2. In the Migration Definition Window, enter the details for the following:
   - **Migration Name**: Enter the code of the export of objects to be migrated definition.

     This is a unique identifier.
   - **File Name**: The system auto-creates the file name of the objects that can be used to export the definition in the following format:
     - **For Business Objects**: Migration Name_BO_Time Stamp (MMDDYYY HHMMSS)
     - **For Identity Objects**: Migration Name_IDM_Time Stamp (MMDDYYY HHMMYY)
3. Click **Apply** to save the details.

   The Object Selection Page is displayed.
4. In the Object Selection Page, select the required Object Type from the Object Types drop-down list.

   The object types listed for the System Configuration tab are, Schedule, Batch group, Batch, Pipeline, Threshold, Job and PMF process. For more information about the object types, refer Migration Object Types

   You can also enter the first few letters in Search to add a particular object from a selected Object type.

   The list of objects is displayed.
5. Select the objects to be added to the Migrate Definition.

   The selected objects are added to the respective object type branch.
6. Click **Save** to create the Migrate Definition.

A confirmation message is displayed, when the definition is saved successfully.

The new migration definition is listed in the Object Export Summary Page and the status is set to Saved.

> **NOTE** If the migration definition object is not created successfully and the status is set to Failed. Contact My Oracle Support of more information.

11.3.3 Viewing Migration Export Objects

You can view and edit the migration objects from the **Object Export Summary** Page.

To view the list of migration export objects associated with a migration definition, highlight the migration definition and click **Menu** button  .

The following options are displayed.

Field	Description
View Log	View the migration log details of the selected Migration Definition. For more information, refer to View Log of Migration Export Objects.
View	View the Object Details for a specific Migration Definition. Click an object to view more details. For more information, refer to Viewing Export Migration Object Details.
Export	Click Export to initiate Object Migration (Export) for a specific Migration Definition. When the migration is completed, the status will change from Saved to Success. For more information, refer to Exporting Migration Objects.
Edit	Click Edit to view and edit the objects linked to a Migration Definition. For more information, refer to Editing Migration Export Definitions.

11.3.3.1 View Log of Migration Export Objects

To view the log details of object with migration status Success or Failed, follow these steps. The view log facilitates you to view the log information of the definition for export of objects to be migrated with its status.

1. Highlight the migration definition and click **Menu** button  and select **View Log**.

   The View Log Page is displayed.

   The export migration status with the following details is displayed.

Field	Description
Object Migration ID	The migration ID associated with the export object.
Object Type	The object type of the export object.

Field	Description
Object Code	The object code associated with the export object.
Creation Date	The date of creation of the export object.
Created By	The User Id of the User who created the export object.
Status	The migration status of the export object. • Success - Indicates that the export migration was completed successfully. • Failed - Indicates that the export migration did not complete

> **NOTE** The View Log Page for a migration object with status Saved will be empty

2. Click **OK** to close the window.

11.3.3.2 Viewing Export Migration Object Details

To view the list of objects added to a specific Migration Definition:

1. Highlight the migration definition and click **Menu** button  and select **View**.

   The list of migration objects added to the definition is displayed.
2. Double-click an object to view the object attribute details.

11.3.4 Exporting Migration Objects

To export the list of objects added to a specific Migration Definition:

1. Highlight the migration definition and click **Menu** button  and select **Export**.

   After you export, the following types of status are displayed:
   - **Success** - When the object is migrated successfully
   - **Failed** - When the object migration didn't complete.

> **NOTE** Object Migration (export) facilitates you to export a set of objects within the same setup or across different setups

11.3.5 Editing Migration Export Definitions

You can edit the migration export objects that are not exported and their status is Saved or Failed. If the object is already exported and the status is set to Success, you cannot edit the object details.

To edit a record of the definition of export of objects to be migrated, follow these steps. You can add more objects to export or remove existing objects.

1. Highlight the migration definition and click **Menu** button  and select **Edit**.

The Object Selection Window is displayed.

2. Update the required details.
3. In the Object Selection window, select the required Object Type from the Object Types drop-down list.

   The object types listed for the System Configuration tab are, Batch_Group, PMF_Process, Batch and Schedule. For more information about the object types, refer Migration Object Types.

   You can also enter the first few letters in Search to add a particular object from a selected Object type.

   The list of objects is displayed.
4. Select the objects to be added to the Migrate Definition or to be deleted from the Migration Definition.

   The selected objects are added/deleted to the respective object type branch.
5. Click **Save** to edit the Migrate Definition.

   A confirmation message is displayed, when the definition is saved successfully.

   The edited migration definition is listed in the Object Export Summary Page and the status is set to Saved.
6. Click **Save** to update the changes.

11.3.6 Deleting Migration Export Objects Definition

You can delete only a record that is set to Saved or Failed status and not a record that is in Success status.

To delete a migrate export object definition, follow these steps.

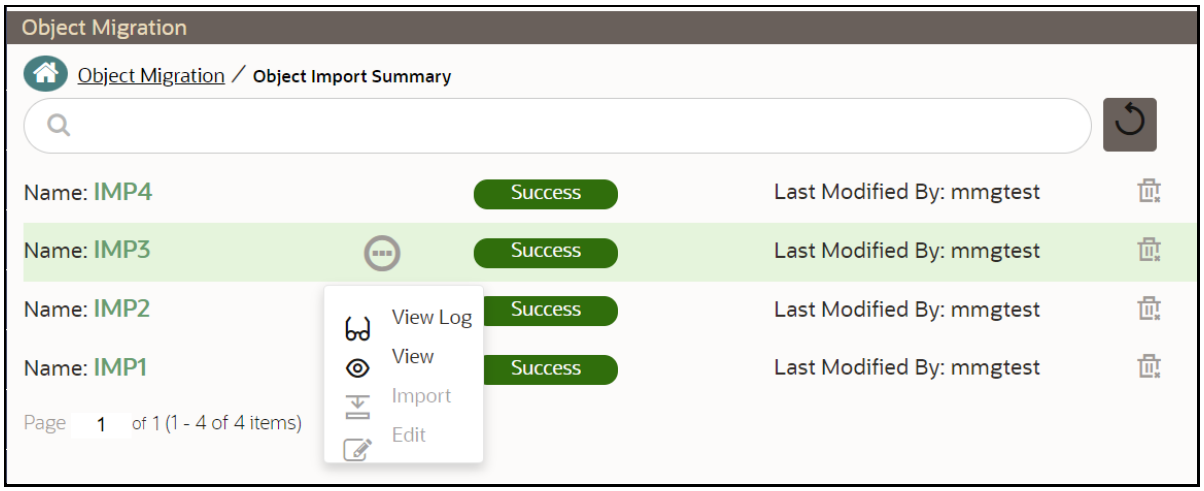
1. Highlight the record to be deleted and click the **Delete** button.
2. Click **Yes** to confirm and proceed with the deletion.

11.4 Object Import Summary

To access the Object, Import Summary page, follow these steps:

1. Click **Launch Workspace** next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. Click the Object Migration icon and select **Import Objects**.

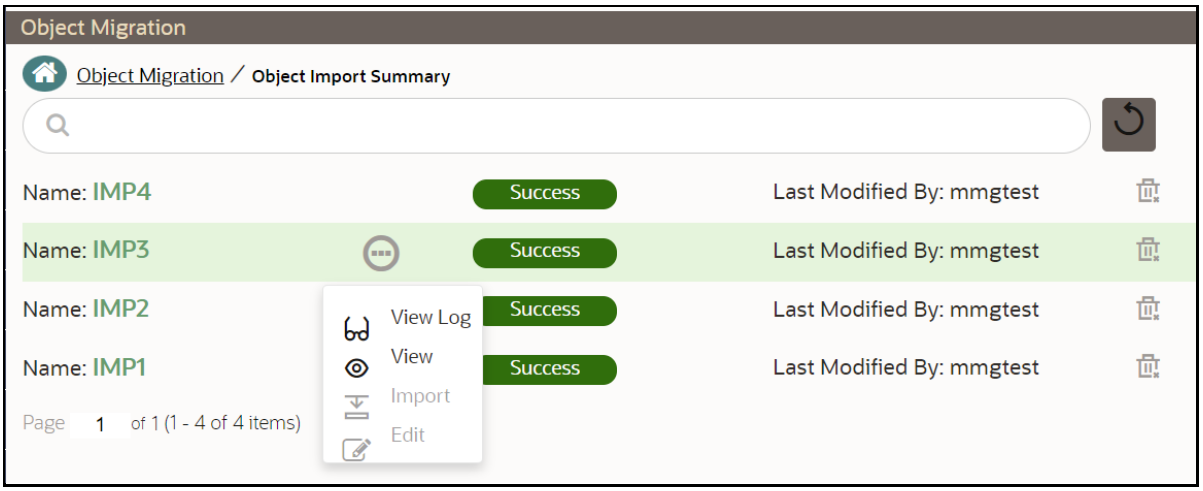
   The Object Import Summary page containing the records is displayed with the following details.



- **Name:** The unique migration name assigned to the collection when the migration definition was created.
- **Object Migration Status:** The migration status of the record corresponding to the specified Definition Name. The three migration status values are as follows:
  - **Success** - Set to Success, when the object import is completed successfully.
  - **Saved** - Set to Saved, when the migration definition is ready for import and needs to import.
  - **Failed** - Set to failed, when the migration definition is not imported successfully.
- **Last Modified By**: The ID of the Last Modified by user who has modified the record.

11.4.1 Navigating Migration Object Import Summary Page

To search for a specific migration definition, type the first few letters of the user name that you want to search in the **Search** box and click Search icon. The search results display the names that consist of your search string in the list of available users.



At the bottom of the page, you can enter the number of entries that are available on a single page in the Records box. You can increase or decrease the number of entries that are displayed using the up and down arrows. To navigate between pages in the **View** bar, use the following buttons:

- Use the **First Page** button to view the entries in the first page.
- Use the **Previous Page** button to view the entries in the previous page.
- Use the **Next Page** button to view the entries in the next page.
- Use the **Last Page** button to view the entries in the last page.

You can also navigate to the desired page. To do this, enter the page number in the **View** bar control and press **Enter**.

To create a new migration definition, click **Add** button and proceed with Creating Migration Import Object Definitions.

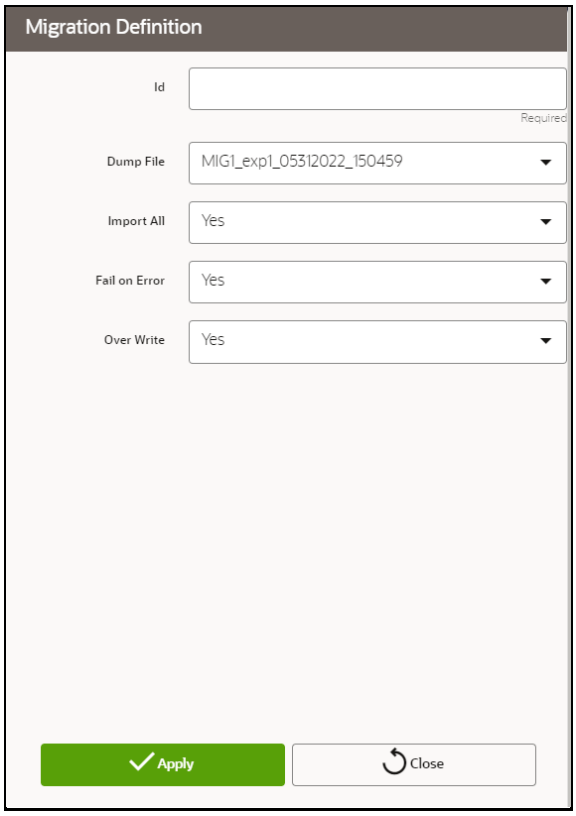
11.4.2 Creating Migration Import Object Definitions

You can create Migration Import object definitions for the following object types.

- Batch Group
- Threshold
- PMF Process
- Batch
- Pipeline
- Schedule
- Job

To create a definition for import of objects to be migrated, do as follows:

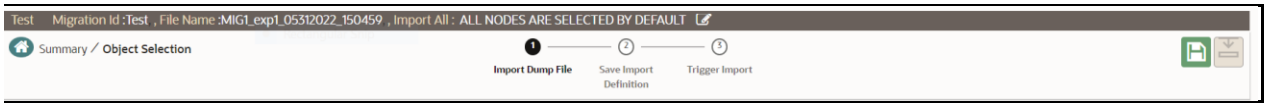
1. Click **Add** in the **Object Import Summary** Page to display the **Migration Definition Window**.



2. In the **Migration Definition** Window, enter the details for the following:
   - **ID**: Enter a valid name for the new migration import definition.

     This is a unique identifier.
   - **Dump File:** Select the dump file to be utilized for creating the Migration Import Definition.
3. **Import All:** Select an option to import the nodes that are associated with the selected object type

   You can edit this option if required, in the **Object Selection** page.



   - **Yes** - Imports all the nodes that are included in the dump file.
   - **No** - Imports only those nodes that you can select in the **Object Selection** page.
4. **Fail on Error:** Select an option to proceed with the definition creation in case of an error.

   You can edit this option if required, in the **Object Selection** Page.
   - **Yes** - Stops the creation process, if error is generated
   - **No** - Creates the import definition even when error is generated. The node with the error is not included in the object creation.
5. **Overwrite:** Select an option to overwrite the existing definition file.

   You can edit this option if required, in the **Object Selection** Page.

- **Yes** - Replaces the existing Import definition.
- **No** - Creates a new Import definition.

6. Click **Apply** to save the details.

   The **Object Selection** page is displayed.

7. In the **Object Selection** page, click Add Members, to add objects associated with specific object types.

8. In the **Object Selection** page, select the required **Object Type** from the **Object Types** drop-down list.

   The object types listed are, Schedule, Batch group, Batch, Pipeline, Threshold, Job and PMF process. For more information about the object types, refer Migration Object Types.

   You can also enter the first few letters in **Search** to add a particular object from a selected Object type.

   The list of objects is displayed.

9. Select the objects to be added to the Migrate Definition.

   You can select the objects only if you select **No** for **Import All**.

   The selected objects are added to the respective object type branch.

10. Click **Save** to create the Migrate Definition.

    A confirmation message is displayed, when the definition is saved successfully.

    The new migration definition is listed in the Object Import Summary Page and the status is set to **Saved**.

> **NOTE** If the migration definition object is not created successfully and the status is set to Failed. Contact My Oracle Support of more information.

11.4.3 Viewing Migration Import Objects

You can view and edit the migration objects from the **Object Import Summary** Page.

To view the list of migration export objects associated with a migration definition, highlight the migration definition and click **Menu** button  .

The following options are displayed.

Field	Description
View Log	View the migration log details of the selected Migration Definition. For more information, refer to View Log of Migration Import Objects.
View	View the **Object Details** for a specific Migration Definition. Click an object to view more details. For more information, refer to Viewing Import Migration Object Details.

Field	Description
Export	Click **Export** to initiate Object Migration (Import) for a specific Migration Definition. When the migration is completed, the status will change from **Saved** to **Success**. For more information, refer to Importing Migration Objects.
Edit	Click **Edit** to view and edit the objects linked to a Migration Definition. For more information, refer to Editing Migration Import Definitions.

11.4.3.1 View Log of Migration Import Objects

To view the log information of a record of the definition for import of objects to be migrated, follow these steps. The view log facilitates you to view the log information of the definition for import of objects to be migrated with its status.

1. Highlight the migration definition and click **Menu** button  and select **View Log**.

   The View Log Page is displayed.

   The export migration status with the following details is displayed.

Field	Description
Object Migration ID	The migration ID associated with the export object.
Object Type	The object type of the export object.
Object Code	The object code associated with the export object.
Creation Date	The date of creation of the export object.
Created By	The User Id of the User who created the export object.
Status	The migration status of the export object. • Success - Indicates that the export migration was completed successfully. • Failed - Indicates that the export migration did not complete

> **NOTE** The View Log Page for a migration object with status Saved will be empty

2. Click **OK** to close the window.

11.4.3.2 Viewing Import Migration Object Details

To view the list of objects added to a specific Migration Definition:

1. Highlight the migration definition and click **Menu** button  and select **View**.

   The list of migration objects added to the definition is displayed.

2. Double-click an object to view the object attribute details.

11.4.4 Importing Migration Objects

To export the list of objects added to a specific Migration Definition:

1. Highlight the migration definition and click **Menu** button  and select **Export**.

   After you export, the following types of status are displayed:

   - **Success** - When the object is migrated successfully
   - **Failed** - When the object migration didn't complete.

   NOTE Object Migration (export) facilitates you to export a set of objects within the same setup or across different setups

11.4.5 Editing Migration Import Definitions

You can edit the migration export objects that are not exported and their status is Saved or Failed. If the object is already exported and the status is set to Success, you cannot edit the object details.

To edit a record of the definition of export of objects to be migrated, follow these steps. You can add more objects to export or remove existing objects.

1. Highlight the migration definition and click **Menu** button  and select **Edit**.

   The Object Selection Window is displayed.

2. Update the required details.
3. In the Object Selection window, select the required Object Type from the Object Types drop-down list.

   The object types listed for the System Configuration tab are, Batch_Group, PMF_Process, Batch and Schedule. For more information about the object types, refer Migration Object Types.

   You can also enter the first few letters in Search to add a particular object from a selected Object type.

   The list of objects is displayed.

4. Select the objects to be added to the Migrate Definition or to be deleted from the Migration Definition.

   The selected objects are added/deleted to the respective object type branch.

5. Click **Save** to edit the Migrate Definition.

   A confirmation message is displayed, when the definition is saved successfully.

   The edited migration definition is listed in the Object Export Summary Page and the status is set to Saved.

6. Click **Save** to update the changes.

11.4.6 Deleting Migration Import Objects Definition

You can delete only a record that is set to Saved or Failed status and not a record that is in Success status.

To delete a migrate export object definition, follow these steps.

1. Highlight the record to be deleted and click the **Delete** button.
2. Click **Yes** to confirm and proceed with the deletion.

12 Model Training

The Model Training page allows you to add and manage the Model Technique, Model Library, and Model Objectives. You can either import the models from external sources or create, train and label it as champion model.

12.1 Model Objective

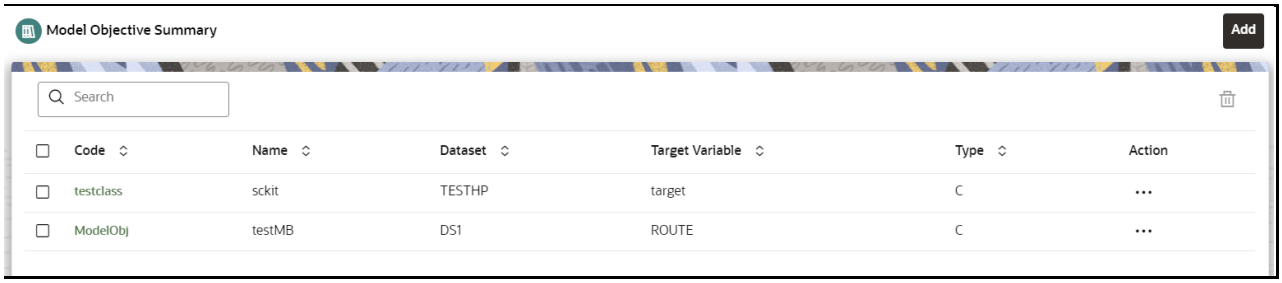
Model Objective is the top level metadata where further models are created or trained which can be consumed in the upcoming steps.

To add a Model Objective , follow these steps:

1. Click **Launch Workspace** next to corresponding Workspace to Launch Workspace to display the **MMG Dashboard** window with application configuration and model creation menu.
2. In the LHS menu, click **Model Training** to display the **Model Objective Summary** window.

   This window displays the model objectives in a table.

Figure 54: Model ObjectiveSummary Page



The following table provides descriptions for the fields and icons on the **Model Objective Summary** page.

Field or Icon	Description
Search	The field to search for Model Objectives. Enter specific terms in the field for which you want to search, and press Enter on the keyboard to display the results.
Code	The unique identifier of the Model Objective.
Name	The name of the Model Objective.
Dataset	The dataset which is used to create the Model Objective.
Target Variable	Target Variable is the important parameter after the dataset. During the dataset selection, target variable is selected that constitutes of number of columns/features.
Type	The type which is selected during model objective creation. The type can be either Classification or Regression.

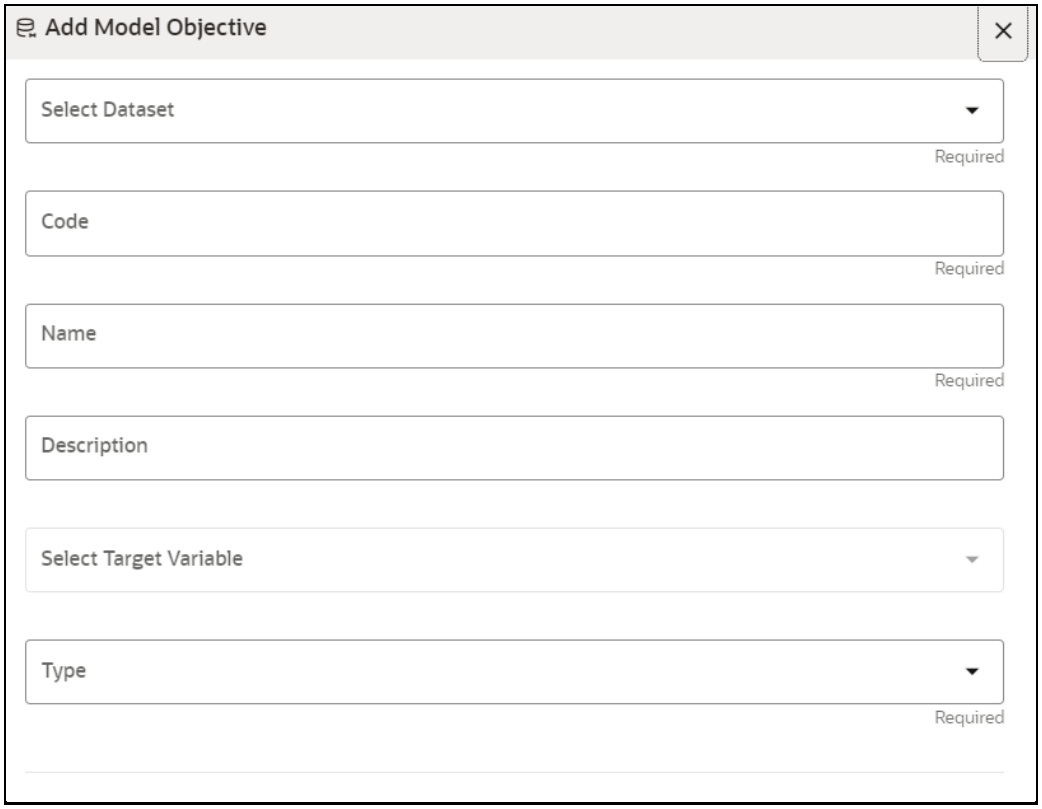
Field or Icon	Description
Action •••	Click the three dots to perform View/Delete functions on the selected model objective..

12.1.1 Adding a Model Objective

To add a model objective, follow these steps:

1. In the **Model Objective Summary** screen, click **Add**.

   The Add Model Objective screen is displayed.



2. Select the required dataset from the drop-down.

   For more details on the dataset, see Dataset section.

3. Enter the code for the Model Objective.

   The code should be unique for each model objective.

4. Enter the name and description for the Model Objective.

5. Select one of the column/feature as the target variable for the models which needs to be trained/uploaded.

6. Select the Type as either as **Classification** or **Regression.**

   Classification: Classification is a process of finding a function which helps in dividing the dataset into classes based on different parameters. The task of the classification algorithm is to find the mapping function to map the input(x) to the discrete output(y).

Regression: Regression is a process of finding the correlations between dependent and independent variables. The task of the Regression algorithm is to find the mapping function to map the input variable(x) to the continuous output variable(y).

7. Click **Create**.

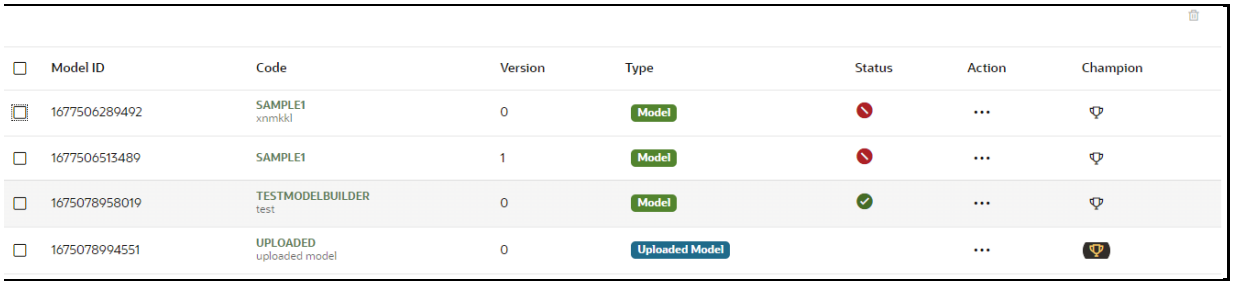
   The Model Objective is created and displayed in the **Model Objective Summary** screen.

12.1.2 View a Model Objective

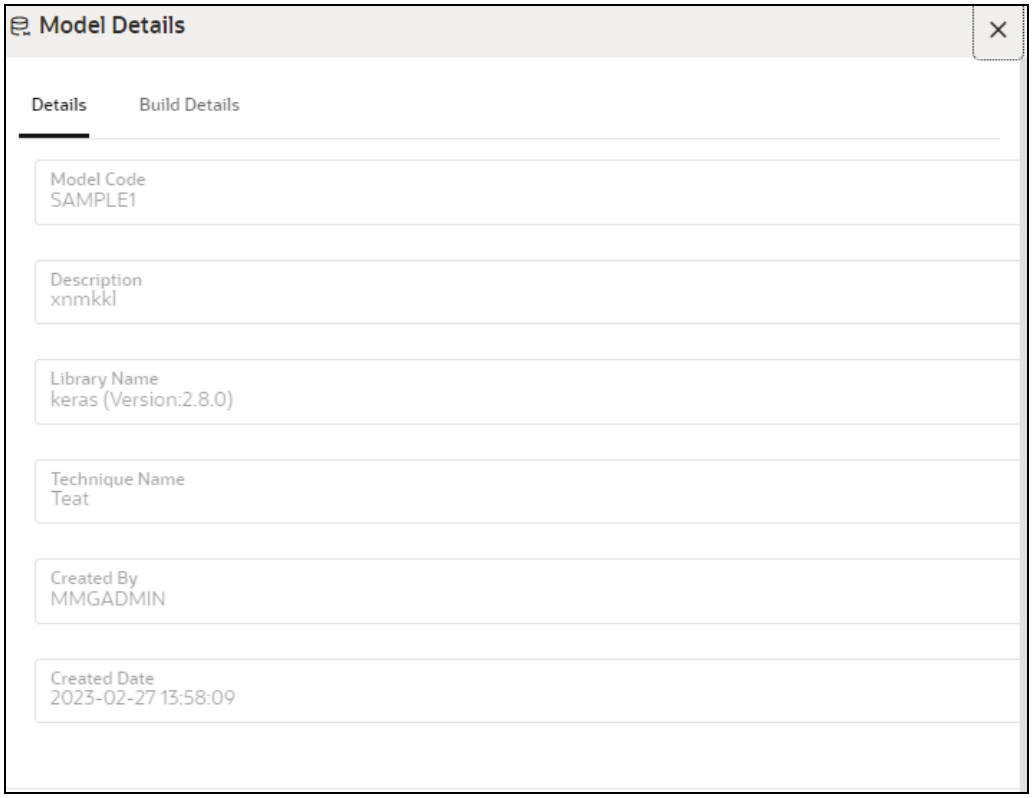
To view a Model Objective, follow these steps:

1. Navigate to **Model Objective Summary** page.
2. Click ••• next to corresponding Model Objective and select View  .

   The following page is displayed.



Clicking on the Type of the model displays the individual model and build details.

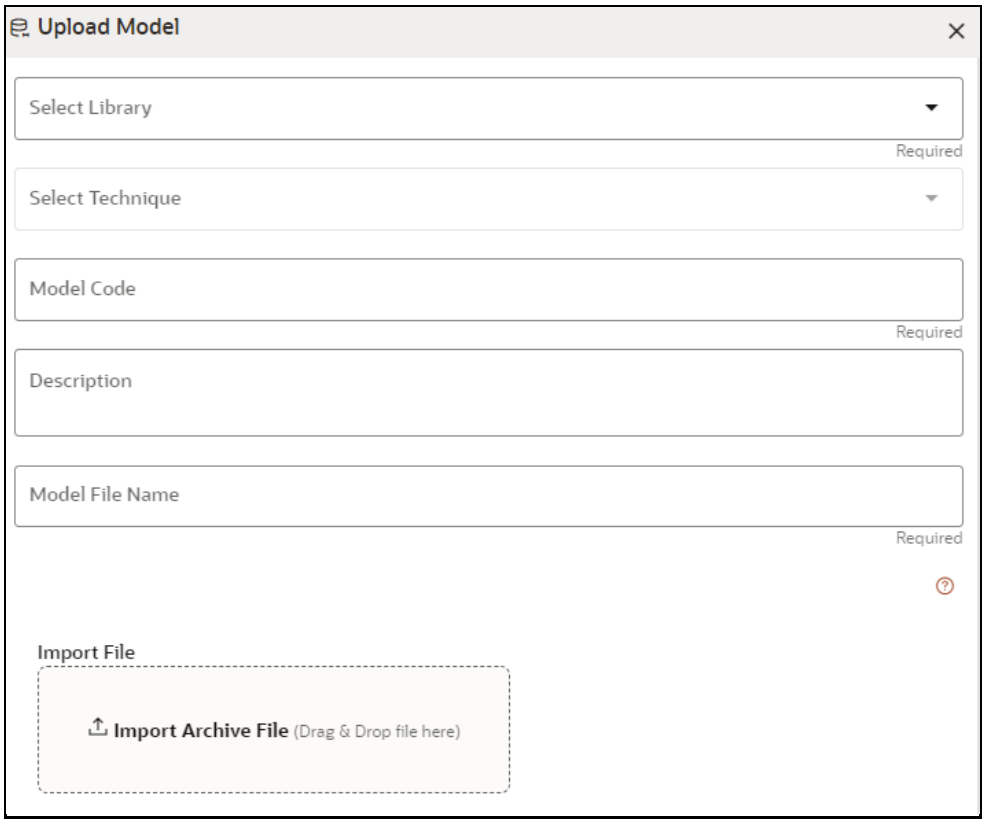


12.1.2.1 Upload a Model

To upload a model, follow these steps:

1. In the View Model Objective page, click **Add** and select **Upload Model**.

   The **Upload Model** page is displayed.



2. Select the library from the drop-down.

   For more details on the model library, see Model Library section.

3. Select the required technique from the drop-down.

   For more details on the model technique, see Model Technique section.

4. Enter the model code and description for the Model Objective.
5. Enter the name of model file with extension for use during Save and Load.

   OR

   Import the model using **Import Archive File** option.

6. Click **Upload Model**.

   The model is uploaded successfully.

12.1.2.2 Build a Model

Building a model consists of the following steps:

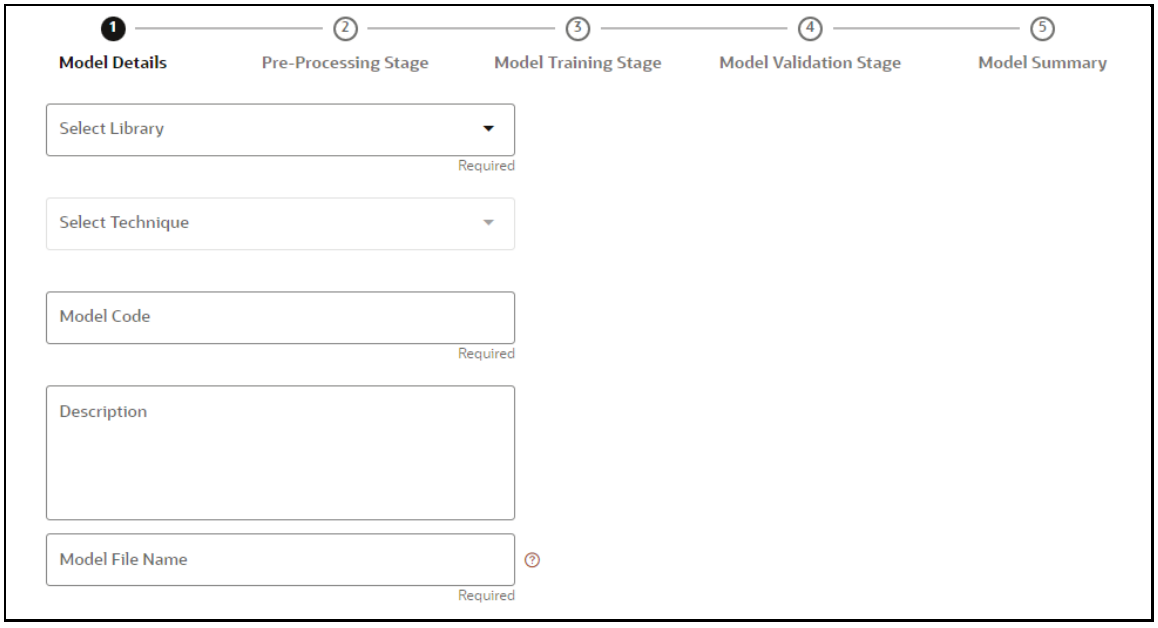
- Model Details

- Pre – Processing Stage
- Model Training Stage
- Model Validation Stage
- Model Summary

12.1.2.2.1 Model Details

1. In the View Model Objective page, click **Add** and select **Build Model**.

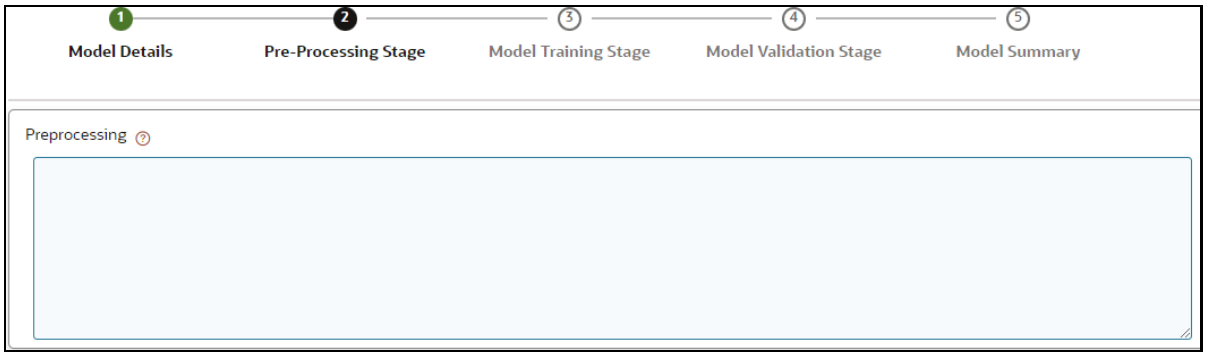
   The **Model Details** page is displayed.



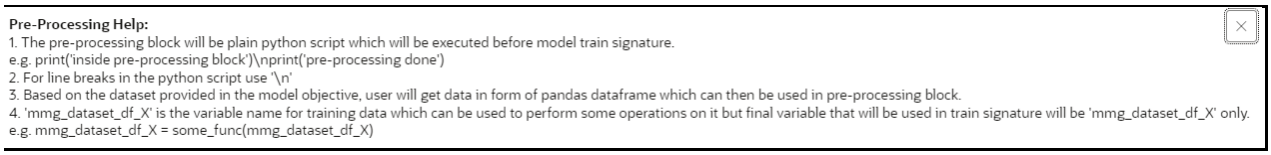
2. Select the library from the drop-down. Currently, the supported libraries are keras, ONNX, scikit-learn, and xgboost. For more details, see Model Library section.
3. Select the required technique from the drop-down. For more details, see Model Technique section.
4. Enter the model code and description for the model.
5. Enter the name of model file with extension for use during Save and Load.
6. Click **Next** to go to the next step.

12.1.2.2.2 Pre – Processing Stage

After adding the details in the Model Details page, the Pre-Processing Stage screen is displayed.



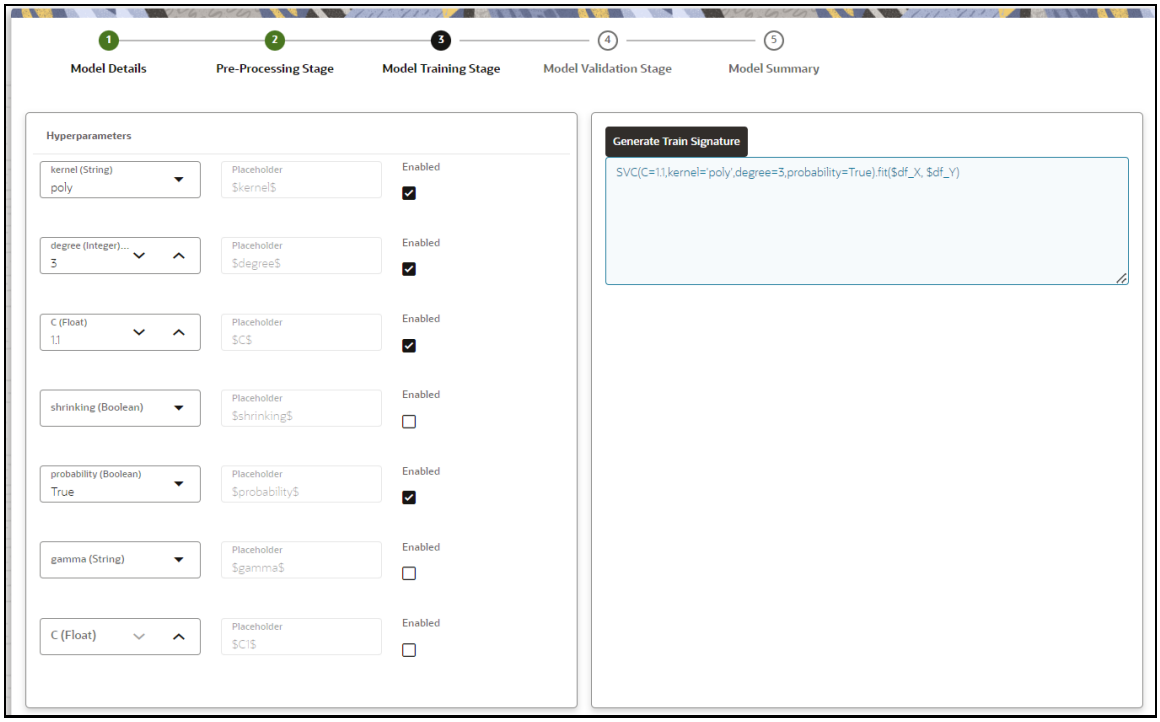
1. Enter the data in the Preprocessing textbox based on the following example.



2. Click **Next** to go to the next step.

12.1.2.2.3 Model Training Stage

After adding the details in the Pre-Processing Stage page, the Model Training Stage screen is displayed.



1. Select the hyperparameters.

   When the parameters is selected, the **Enabled** check box is selected by default.

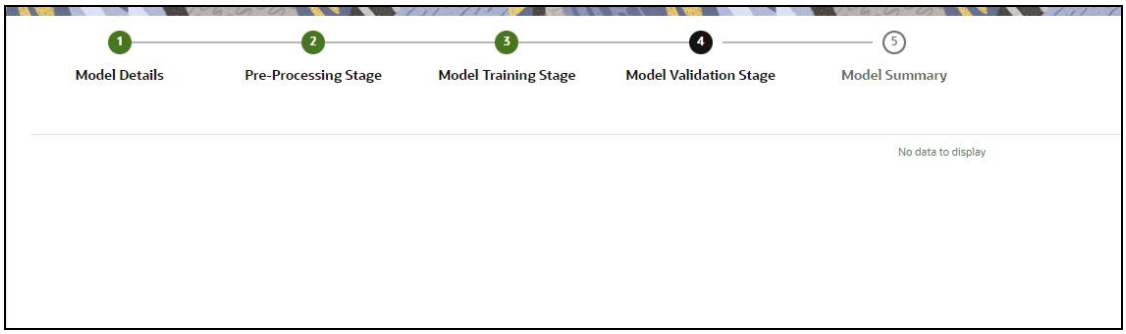
2. Click **Generate Train Signature**.

The signature is displayed.

3. Click **Next** to go to the next step.

12.1.2.2.4 Model Validation Stage

After adding the details in the Model Training Stage page, the Model Validation Stage screen is displayed.

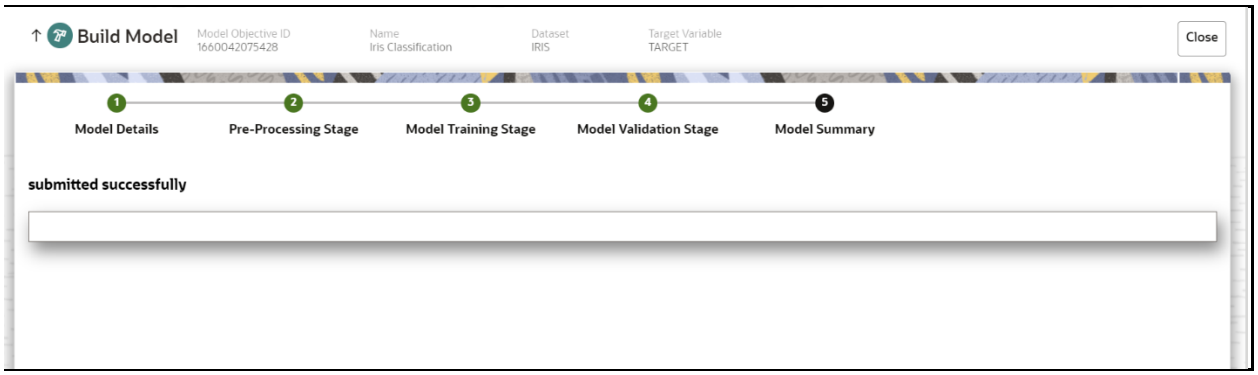


The Validation is work in progress.

1. Click **Finish** to start the training of the model.

12.1.2.2.5 Model Summary

Once the validation of the model is completed, the Model Summary screen is displayed



1. Click **Close**.

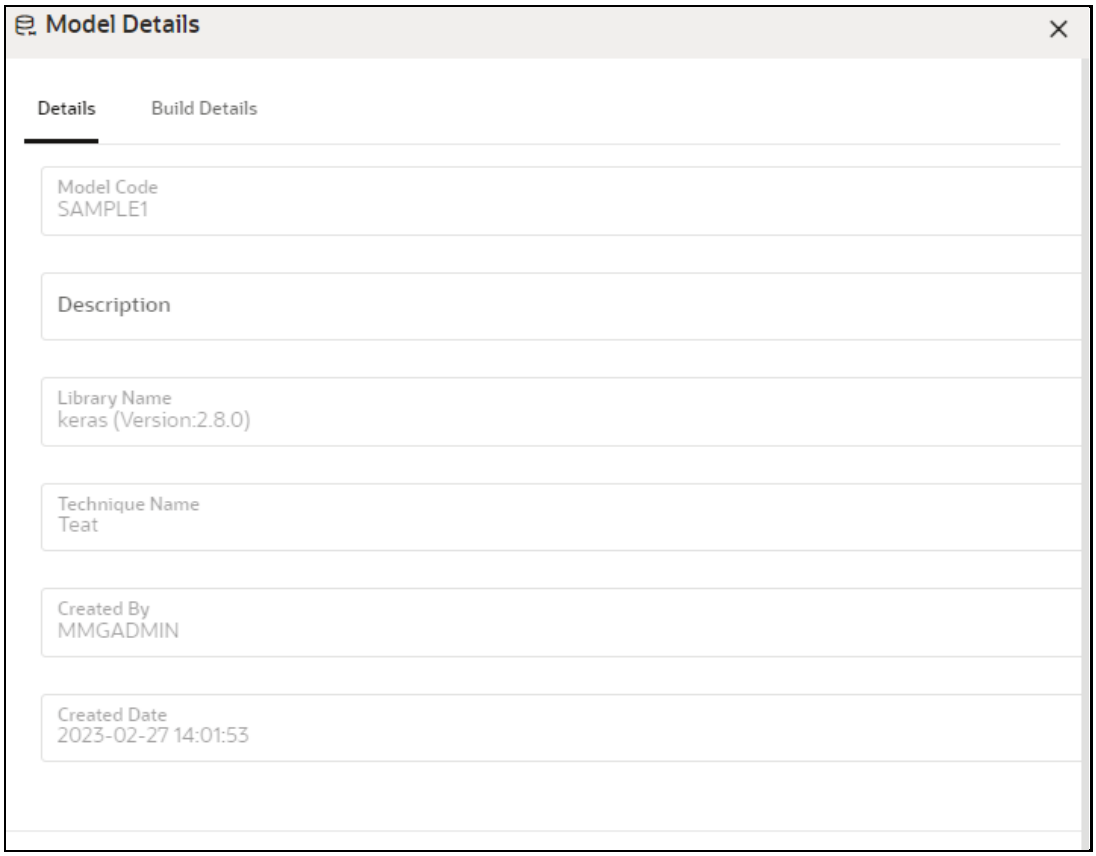
   The model is created and displayed in the Model Objective screen.

12.1.2.3 View Details of a Model

To view details of a model, follow these steps:

1. In the View Model Objective page, click **Action** ••• next to corresponding model and select **View Details**.

The model details are displayed.



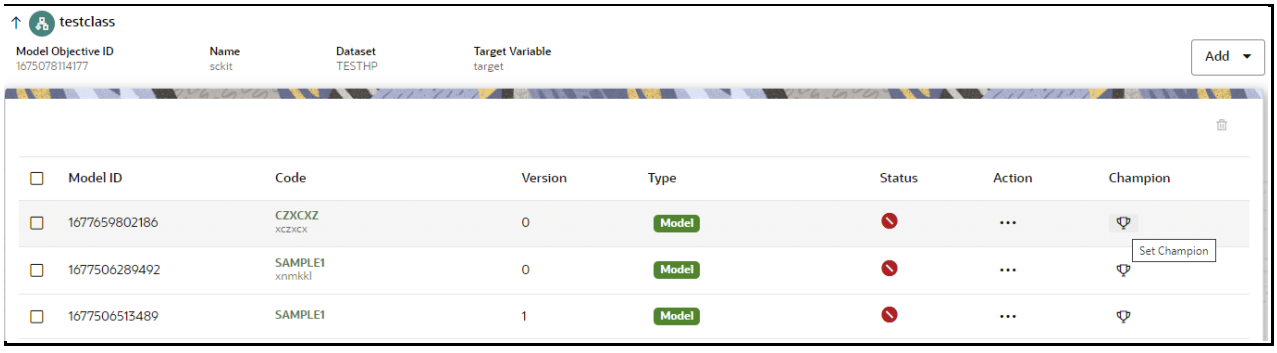
2. Click on the Build Details tab to view the Preprocessing, and Hyperparameters details of the model.

12.1.2.4 Setting the Model as Champion

You can set the trained models as champion model.

To set the model as champion, follow these steps:

1. In the View Model Objective page, hover over the model which you want to set as champion and click **Set Champion** icon.



A confirmation dialog is displayed.

2. Click **Update**.

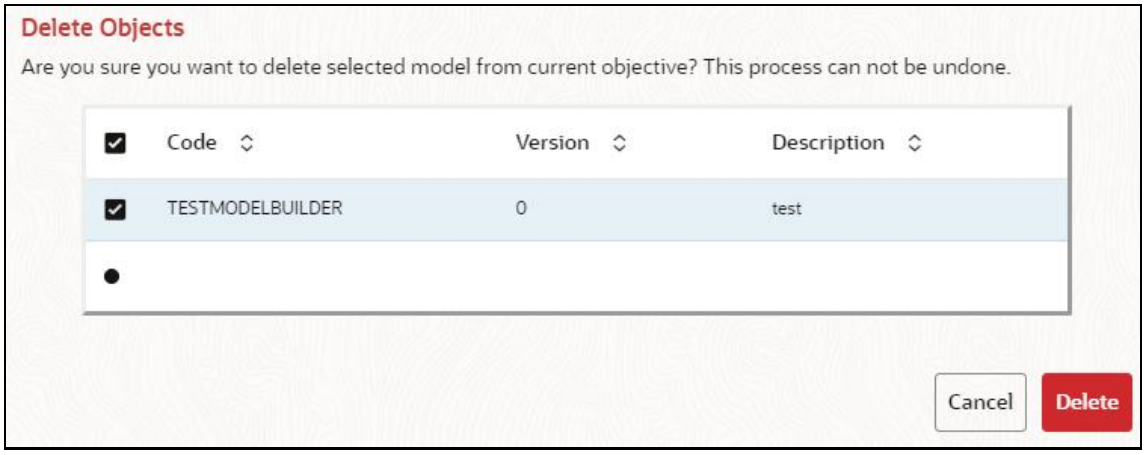
   The model is set as champion for the selected model objective.

12.1.2.5 Delete a Model

To delete a model, follow these steps:

1. In the View Model Objective page, select the models which you want to delete.
2. Click Delete .

   A confirmation dialog is displayed.



3. Click **Delete**.

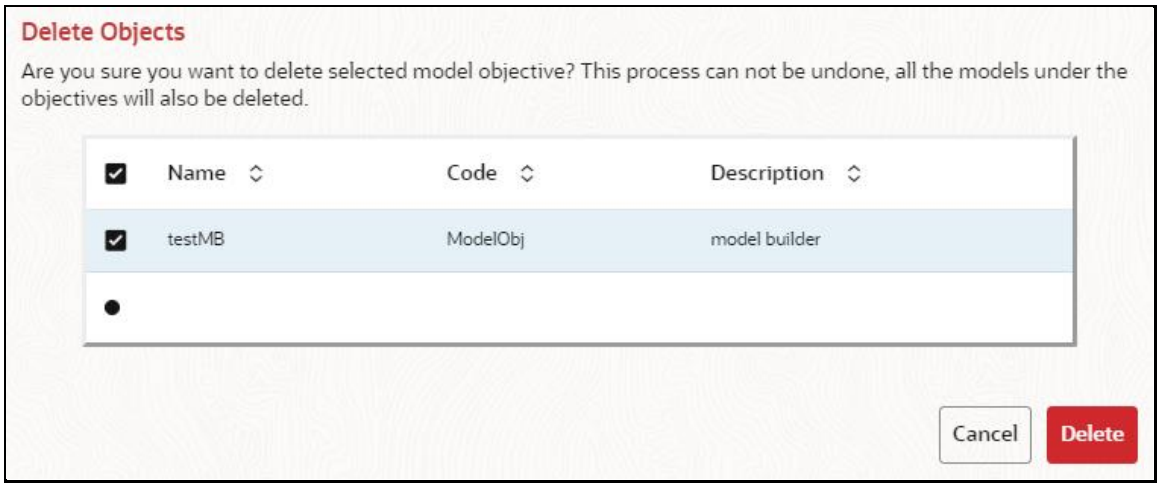
   The selected models are deleted.

12.1.3 Delete a Model Objective

To delete a Dataset, follow these steps:

1. Navigate to Model Objective Summary page.
2. Click ••• next to corresponding Dataset and select Delete .

A confirmation dialog is displayed.



3. Click **Delete**.

   The selected model objective and all the models associated are deleted.

12.2 Model Library

The Model Library information is used to bind a particular technique and its details to one unique Model Library. The Starting point for the Model builder is to register a model library with MMG application after it is properly set up in the python environment to be used.

Currently, MMG supports the following libraries:

- keras
- ONNX
- scikit-learn
- xgboost

As an example, the below section provides information on how to set up and register scikit-learn library.

Setting up the python library in the python environment:

Open the terminal and install the scikit-learn library using the following command:

```
python3 -m pip install scikit-learn
```

> **NOTE** Proxy might be required to install the packages from pip.

Once the installation is complete, check for the package details using the following command:

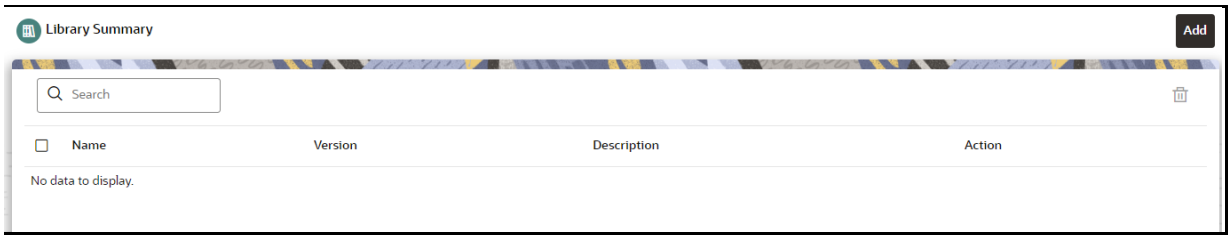
```
pip show scikit-learn
```

Once the installation is complete in python environment, you need to register the library into MMG application.

Registering the python library in MMG:

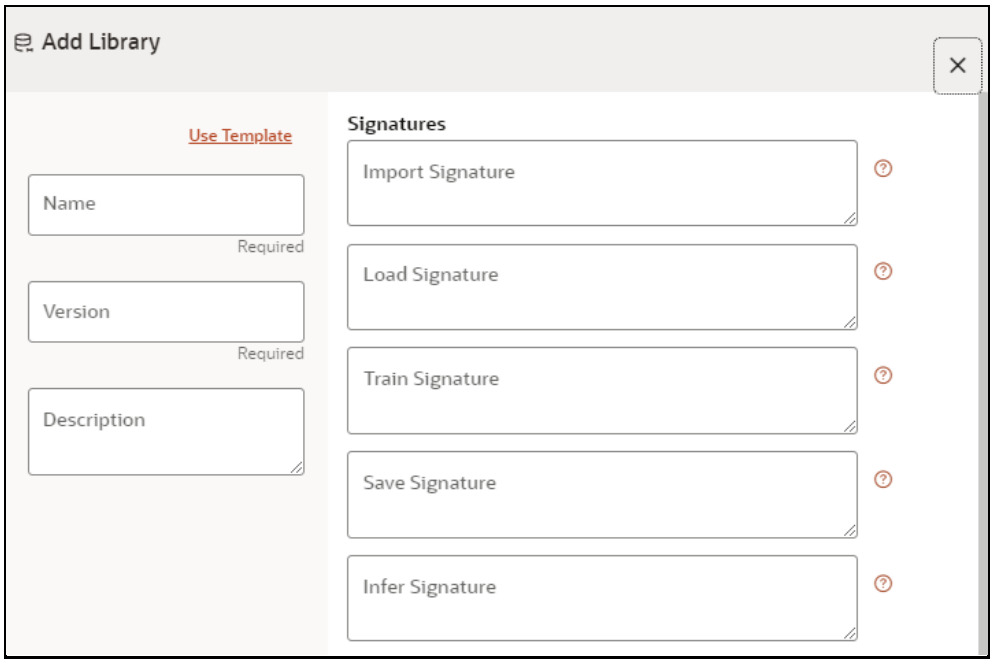
1. In the LHS menu, click **Model Training > Model Library** option.

The **Library Summary** page is displayed.



2. In the **Library Summary** page, click **Add**.

   The **Add Library** page is displayed.



3. Enter the name of the library.
4. Enter the version and description of the library.
5. Enter the signatures such as Import, Load, Train, Save, and Infer. For more details on the signatures, click on the respective help icon.

   Some of the signatures captured in library stage might not be standard across different algorithm/techniques provided by the library.

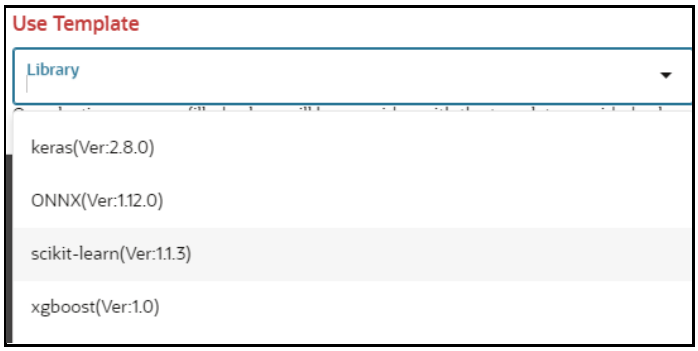
6. Click **Create**.

   The Model Objective is created and displayed in the Model Objective screen.

> **NOTE** MMG User needs to check the details provided above from the homepage of the library being captured.
>
> For example: `https://scikit-learn.org/stable/.`

OR

You can select the **Use Template** option to pre-fill the entries from the seeded list of Libraries. Currently following are the library present in the template.



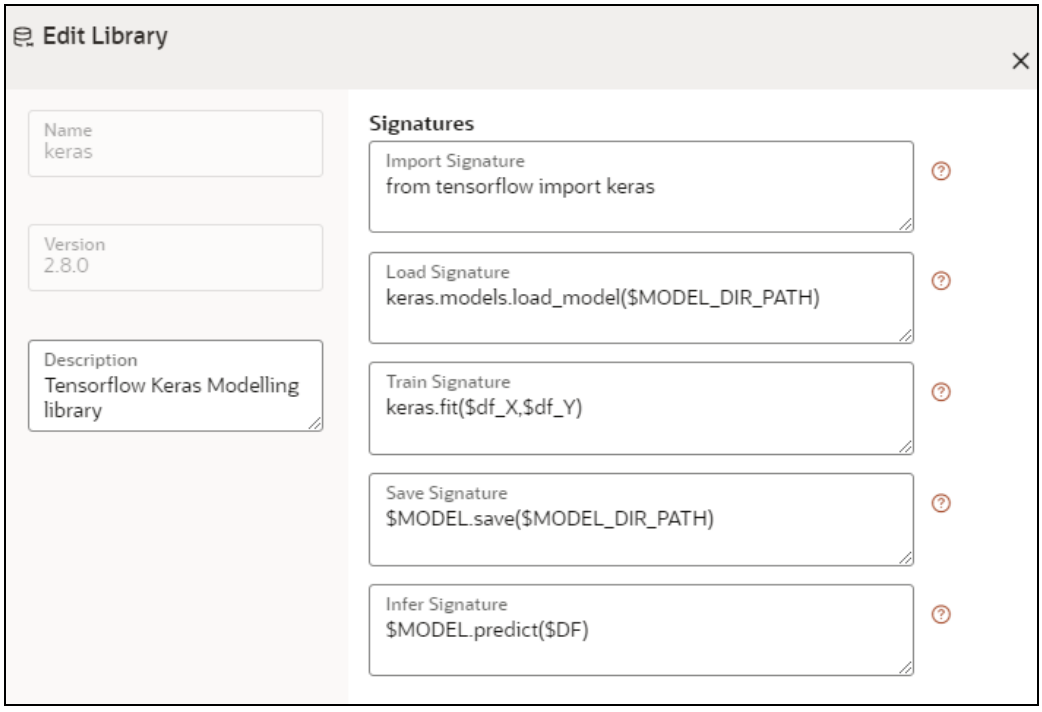
You can also edit the data based on your requirements.

12.2.1 Edit a Model Library

To edit a model library, follow these steps:

1. Navigate to **Library Summary** page.
2. Click ••• next to corresponding model library and select Edit  .

   The **Edit Library** screen is displayed.



3. Make the necessary changes and click **Update**.

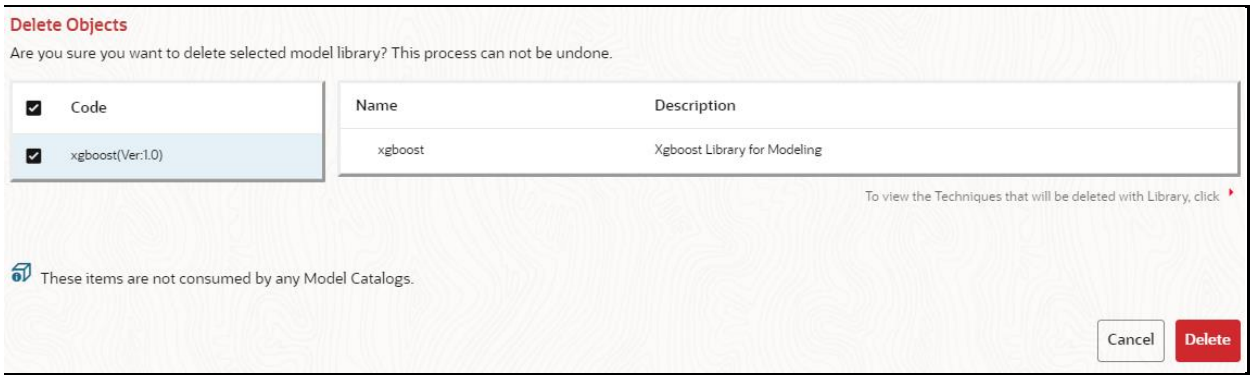
> **NOTE** The Model Library name and version is unique and cannot be modified.

12.2.2 Delete a Model Library

To delete a model library, follow these steps:

1. Navigate to **Library Summary** page.
2. Click ••• next to corresponding model library and select Delete  .

   A confirmation dialog is displayed.



3. Click **Delete**.

   The selected model library and all the model techniques associated are deleted. The libraries that are deleted cannot be consumed by any model catalogs.

   To delete multiple model libraries:

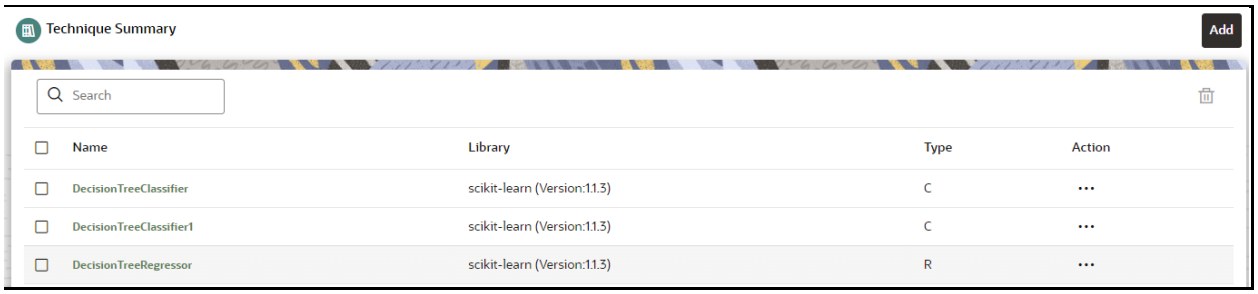
   Select the model libraries which you want to delete and click  on the header of the page.

12.3 Model Technique

Model Technique is the algorithm/technique used to create python model using the library/package which was created in the Model Library Screen. It is the actual information captured in the MMG application that helps in training the model (Upload and Build).

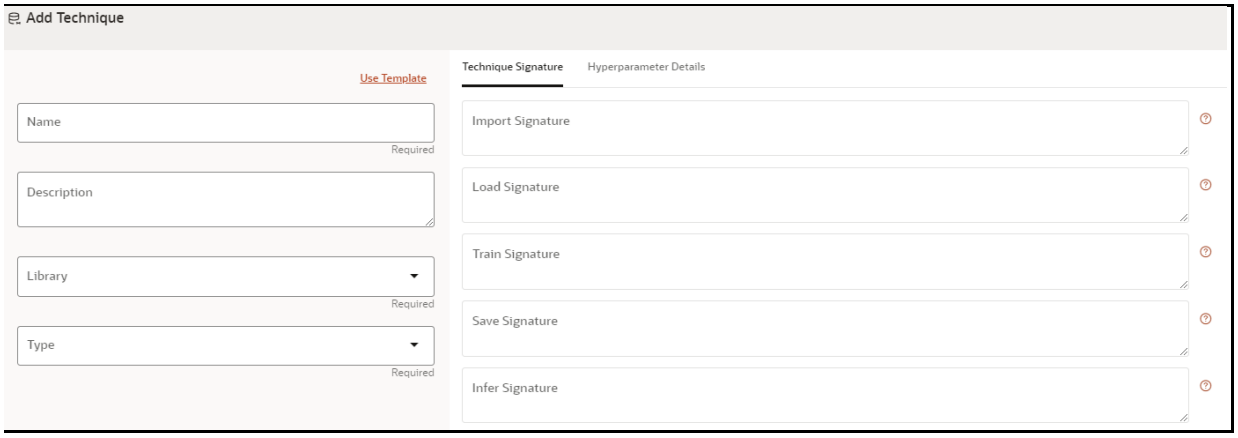
1. In the LHS menu, click **Model Training > Model Technique** option.

The **Technique Summary** page is displayed.



2. In the **Technique Summary** page, click **Add**.

   The **Add Technique** page is displayed.



3. Enter the name of the technique.
4. Enter the description of the technique.
5. Select the library from the drop-down. Currently, MMG supports the libraries such as keras, ONNX, scikit-learn, and xgboost.
6. Select the type as either as **Classification** or **Regression.**

   Classification: Classification is a process of finding a function which helps in dividing the dataset into classes based on different parameters. The task of the classification algorithm is to find the mapping function to map the input(x) to the discrete output(y).

   Regression: Regression is a process of finding the correlations between dependent and independent variables. The task of the Regression algorithm is to find the mapping function to map the input variable(x) to the continuous output variable(y)
7. Enter the signatures such as Import, Load, Train, Save, and Infer. For more details on the signatures, click on the respective help icon.

   Some of the signatures captured in library stage might not be standard across different algorithm/technique provided by the library.
8. Navigate to **Hyperparameter Details** tab and click **Add** to add the parameters.

   You can add the type of parameters such as String, Integer, Float, and Boolean.

9. Click **Create**.

   The Model Technique is created and displayed in the **Technique Summary** screen.

   > **NOTE** MMG User needs to check the details provided above from the homepage of the library being captured.
   >
   > For example: `https://scikit-learn.org/stable/`.

   OR

   You can select the Use Template option to pre-fill the entries from the seeded list of Libraries.

   You can also edit the data based on your requirements.

12.3.1 Edit a Model Technique

To edit a model technique, follow these steps:

1. Navigate to **Technique Summary** page.
2. Click ••• next to corresponding model technique and select Edit .

   The **Edit Technique** screen is displayed.

3. Make the necessary changes and click **Update**.

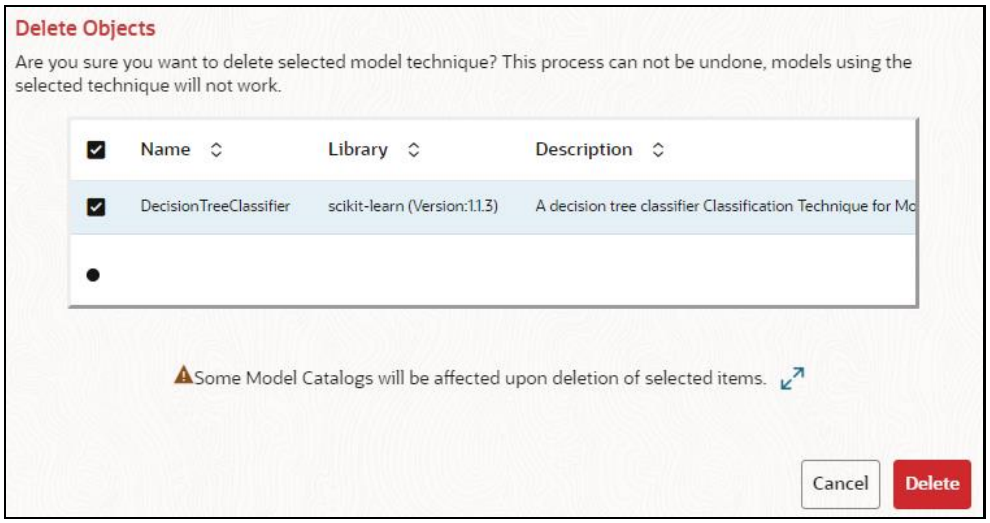
   For more details, see Model Technique section.

12.3.2 Delete a Model Technique

To delete a model technique, follow these steps:

1. Navigate to **Technique Summary** page.
2. Click ••• next to corresponding model technique and select Delete .

A confirmation dialog is displayed.



3. Click **Delete**.

   The selected model technique and all the model catalogs associated are deleted.

   To delete multiple model techniques:

   Select the model techniques which you want to delete and click on the header of the page.

13 Appendix -I

All the listed APIs/Functions can be utilized by users in Model Pipelines.

Table: Python paragraph scripting

API Type	Functionality	Prerequisite	Notebook Execution Script	Notebook Script Input	Minimum Supported Version
Set User	Set the user for a session		%python from mmg.constants import * user = "SAUSER" set_user(user)	user - username to be set	8.1.1
Get User	Get the user that has been set	Set User	%python from mmg.constants import * get_user()		8.1.1
Attach Workspace	Attach the workspace for a session		%python from mmg.workspace import attach_workspace workspace = "CS" attach_workspace(workspace)	workspace- workspace to be attached	8.1.1
Get Workspace	Get the workspace that has been attached	Attach Workspace	%python from mmg.workspace import get_workspace get_workspace()		8.1.1
List Workspaces	Lists all workspaces available to a user	Set User	%python from mmg.workspace import list_workspaces list_workspaces()		8.1.1
Check AIF	Check if AIF is set or not	Set User	%python from mmg.workspace import check_aif check_aif()		8.1.1
List All Data Sources	List all data sources available in a workspace for a set user. The default workspace chosen is the one that is attached.	Set User, Attach Workspace	%python from mmg.workspace import list_datasources list_datasources(order=None)	order - None ensures that all the data sources will be listed for the attached workspace	8.1.1

API Type	Functionality	Prerequisite	Notebook Execution Script	Notebook Script Input	Minimum Supported Version
List All Data Sources of Workspace	List all Data Sources of a specific Workspace that is not the attached workspace for a set user.	Set User	%python from mmg.workspace import list_datasources workspace="HELLO NEW" list_datasources(workspace,order=None)	workspace - Name of workspace for which data sources should be listed order - None ensures that ALL data sources will be listed for that workspace	8.1.1
Fetch First Order Data Source	This method will fetch the data source related to attached works pace with default order 1.	Set User, Attach Workspace	%python from mmg.workspace import list_datasources list_datasources()	order - 1 by default, if not specified	8.1.1
Fetch First Order Data Source of Workspace	This method will fetch the data source with default order 1 related to specified workspace.	Set User	%python from mmg.workspace import list_datasources workspace="CS" list_datasources(workspace)	workspace - Name of workspace for which data source should be listed order- 1 by default if not specified	8.1.1
Fetch Nth Order Data Source	This method will fetch the data source related to attached works pace with given order	Set User, Attach Workspace	%python from mmg.workspace import list_datasources list_datasources(order=2)	order - Order of data source to be fetched	8.1.1

API Type	Functionality	Prerequisite	Notebook Execution Script	Notebook Script Input	Minimum Supported Version
Fetch Nth Order Data Source of Workspace	This method will fetch the data source with given order related to specified workspace.	Set User	%python from mmg.workspace import list_datasources workspace="CS" list_datasources(workspace,order=2)	workspace - Name of workspace for which data source should be listed order = Order of data source to be fetched	8.1.1
Get Connection Object	Returns Oracle/ SQL Alchemy Connection Object to Data Source of order 1 in attached workspace.	Set User, Attach Workspace	%python import cx_Oracle from mmg.workspace import get_connection conn=get_connection() if not isinstance(conn,cx_Oracle.connect): print("Not Connection Object") print(conn)		8.1.2.0
Get Connection Object To Specific Data Source	Returns Oracle/ SQL Alchemy Connection Object to specified Data Source	Set User	%python import cx_Oracle from mmg.workspace import get_connection datasource="DS001" conn=get_connection(datasource) if not isinstance(conn,cx_Oracle.connect): print("Not Conn Object") print(conn)	datasource - Name of datasource for which the connection object has to be returned	8.1.2.0

API Type	Functionality	Prerequisite	Notebook Execution Script	Notebook Script Input	Minimum Supported Version
Get Connection Object To Data Source Using Order	Returns Oracle/ SQL Alchemy Connection Object to Data Source of specified order in attached works pace.	Set User, Attach Workspace	%python import cx_Oracle from mmg.workspace import get_connection datasource=2 conn=get_connection(datasource) if not isinstance(conn,cx_Oracle.connect): print("Not Conn Object") print(conn)	datasource - Order number of datasource in attached workspace for which the connection object has to be returned	8.1.2.0
Test Connection Object	This script can be used to test the connection object to data source	Get Connection Object OR Get Connection Object To Specific Data Source OR Get Connection Object To Data Source Using Order	%python query="SELECT table_name FROM user_tables" cur = conn.cursor() cur.execute(query) print(cur.fetchall()) if cur: cur.close()		8.1.2.0

PYTHON LINEPARSER APIs (for MMG version 8.1.0)

API Type	Functionality	Notebook Execution Script	Note
Attach Workspace	Attach the workspace for a session	%python mmg.attachWorkspace(workspace) #eg- mmg.attachWorkspace("CS") 	workspace - workspace to be attached (String) Sets 'mmg' python variable which contains the information about the attached workspace. Output- Print a boolean in one line on the status of workspace attachment and another line of boolean on status of AIF enabled and attached.

API Type	Functionality	Notebook Execution Script	Note
List Workspaces	Lists all workspaces available to a user	%python mmg.listWorkspaces()	Sets 'ofs_wsList' python variable which is the list of all available workspaces. Output- Print the list of workspaces.
Check AIF	Check if AIF is set or not	%python print(aif__isAttached)	Output- Boolean on the status of AIF.

Get Connection Object	Returns cx_Oracle Connection Object	%python
		mmg.getConnection(schema_name)
		#eg-
		mmg.getConnection("OFSAA")
			schema_name = name of the schema for which connection is required (String).
			Sets 'conn' python variable which is the cx_Oracle connection object to be used for firing queries.

Publish	Fetch and sets the Notebook JSON dump	%python mmg.publish()	Sets 'emf_para' python variable which is the JSON dump for the notebook component. Output- Print the set 'emf_para' value.
Register	Register the Notebook	%python mmg.register(register) #eg- mmg.register({"modelname":"model1",\ # "modeldescription":"Model description"}) #eg- mmg.register(emf_para) {After doing mmg.publish()}	register - JSON Stringify object for RegisterNotebookBean Object. Sets 'publishedNotebookId' python variable which is the studio notebook id for the published version. Output- Prints the published notebook id.
Load Flat File		%python mmg.loadFlatFile() #eg- mmg.loadFlatFile() Incomplete	

Profile Data		%python
		mmg.profileData(code,title)
		#eg-
		mmg.profileData(ABC,XYZ)	

14 Appendix - II

Public APIs for Scheduler Operations

The following APIs are exposed for the Scheduler Operations.

NOTES

1. All the below APIs are POST requests.
2. All the APIs accept below values in request header:
   - ofs_tenant_id - Optional
   - ofs_service_id - Optional
   - ofs_workspace_id - Respective workspace id where the batch needs to be created
   - ofs_remote_user – MMG login user

   The ofs_workspace_id and ofs_remote_user are mandatory fields.

Table: APIs for Scheduler Operations

Base URL: `http(s)://<MMG_Service_HostName>:<MMG-Service_Port>/<CONETXTNAME>/`

Functionality	API	Sample Request JSON	Sample Response JSON	Comments

Execute Immediate	rest-api/v1/external/trigger	{
		"batchName": "batch1",
		"batchType": "rest",
		"includedTasks": "task1,task2",
		"excludedTasks": "task1,task2",
		"heldTasks":"task1,task2"
		}
			{
			"severity": "info",
			"summary": "Object triggered successfully with Run Id: batch1_2022-08-17_1660721567845_1",
			"batchRunId": "batch1_2022-08-17_1660721567845_1",
			"details": "Object triggered successfully.",
			"status": "success",
			"statusCode": "0"
			}
				• The includedTasks/ excludedTasks/ heldTasks are options keys.
				• The batchType will be “group” for batch group execution.

Functionality	API	Sample Request JSON	Sample Response JSON	Comments

Execution Status	rest-api/v1/external/status	{
		
		"batchRunId": "batch1_2022-08-17_1660721567845_1",
		
		"tasks":["task1","task2"]
		}
			{
			"severity": "info",
			"batchRunId": "batch1_2022-08-17_1660721567845_1",
			
			"taskStatusList": [
			{
			
			"taskCode": "task1",
			
			"taskStatus": "SUCCESSFUL",
			
			"statusCode": "0"
			},
			{
			
			"taskCode": "task2",
			
			"taskStatus": "EXCLUDED",
			
			"statusCode": "4"
			}
			],
			
			"batchStatusCode": "0",
			"batchList": [],
			"batchStatus": "SUCCESSFUL",
			"status": "success",
			"statusCode": "0"
			}
				• "tasks" is options key.
				If not passed, then response will contain status all the tasks inside a batch.

Functionality	API	Sample Request JSON	Sample Response JSON	Comments
Rerun	rest-api/v1/external/rerun	{ "batchName": "batchgroup1", "batchRunId": "batchgroup1_2022-08-17_1660720814942_1" }	{ "severity": "info", "summary": "Object triggered successfully for rerun with Run Id: batchgroup1_2022-08-17_1660730049819_1", "batchRunId": "batchgroup1_2022-08-17_1660730049819_1", "details": "Object triggered successfully.", "status": "success", "statusCode": "0" }	

Functionality	API	Sample Request JSON	Sample Response JSON	Comments

Restart	rest-api/v1/external/restart	{
		"batchName": "MMG_R1",
		
		"batchRunId": "MMG_R1_2022-07-15_1657867378160_1"
		}
			{
			"severity": "info",
			"summary": "Object triggered successfully for restart with Run Id: MMG_R1_2022-07-15_1657867378160_1",
			"batchRunId": "MMG_R1_2022-07-15_1657867378160_1",
			"details": "Object triggered successfully.",
			"status": "success",
			"statusCode": "0"
			}	

Functionality	API	Sample Request JSON	Sample Response JSON	Comments

Interrupt	rest-api/v1/external/interrupt	{
		"batchName": "B2001",
		
		"batchRunId": "B2001_2022-05-22_1653222717896_1"
		}
			{
			"summary": "Execution interrupted successfully for Run Id: B2001_2022-05-30_1653233511394_1",
			"severity": "info",
			"batchRunId": "B2001_2022-05-30_1653233511394_1",
			"details": "Execution interrupted successfully.",
			"statusCode": "0",
			"status": "success"
			}	

Failed Response Body

Failed Sample Response JSON	Description

Batch Status Failed	{
	"severity": "info",
	"batchRunId": "BT-BI-EXCHG_RATES_EOD_2022-05-27_1653662703599_1",
	"batchStatusCode": "-1",
	"batchList": [],
	"batchStatus": "FAILED",
	"statusCode": "0",
	"status": "success"
	}

Object Not found	{
	"severity": "error",
	"summary": "Object does not exist.",
	"details": "Object does not exist.",
	"error": {
	"errorCode": "OBJECT_NOT_EXIST",
	"errorMsg": "Object does not exist."
	},
	"statusCode": "-3",
	"status": "failed"
	}

Status Codes in the Response Body

Status Code	Description
0	Success
1	Not Started
2	Ongoing
4	Excluded
5	Held
-1	Failure
-2	Interrupted

Status Code	Description
-3	Object does not exist
-4	Invalid arguments passed in request/not enough parameters in the Request body
-5	Invalid request headers/request headers missing
-6	No executable job is present
-7	Job is already interrupted

OFSAA Support

Raise a Service Request (SR) in My Oracle Support (MOS) for queries related to the OFSAA applications.

Send Us Your Comments

Oracle welcomes your comments and suggestions on the quality and usefulness of this publication. Your input is an important part of the information used for revision.

- Did you find any errors?
- Is the information clearly presented?
- Do you need more information? If so, where?
- Are the examples correct? Do you need more examples?
- What features did you like most about this manual?

If you find any errors or have any other suggestions for improvement, indicate the title and part number of the documentation along with the chapter/section/page number (if available) and contact the My Oracle Support.

Before sending us your comments, you might like to ensure that you have the latest version of the document wherein any of your concerns have already been addressed. You can access My Oracle Support site that has all the revised or recently released documents.