Oracle® Financial Crime and Compliance Management Cloud Service

Using Data Transformation Pipelines

Release 25.08.01
G40990-01
August 2025

Oracle Financial Crime and Compliance Management Cloud Service Using Data Transformation Pipelines, Release 25.08.01

G40990-01

Contents

About This Content

Use this guide to understand how to use the Data Transformation Platform to quickly configure your scenarios to address changes in regulations, adapt to new product introduction, and counter more sophisticated money laundering techniques.

Audience

This guide is intended for users who are responsible for implementing the scenarios in line with the Risk analysis formulated by the compliance team and by Compliance Officers who need to understand the additional possibilities offered by scenarios based on data they are capturing, in order to make their AML program more effective and efficient. Oracle expects these users to be senior compliance officers, who have significant understanding of both their data, the specific behaviors of interest, and the logic required to generate meaningful alerts.

Documentation Accessibility

For information about Oracle's commitment to accessibility, visit the Oracle Accessibility Program website at http://www.oracle.com/pls/topic/lookup?ctx=acc&id=docacc.

Access to Oracle Support

Oracle customers that have purchased support have access to electronic support through My Oracle Support. For information, visit http://www.oracle.com/pls/topic/lookup?ctx=acc&id=info or visit http://www.oracle.com/pls/topic/lookup?ctx=acc&id=trs if you are hearing impaired.

Related Resources

For more information, see Oracle Financial Services Crime and Compliance Management Cloud Service Applications documentation.

Conventions

The following text conventions are used in this document.

Convention	Meaning
boldface	Boldface type indicates graphical user interface elements associated with an action, or terms defined in text or the glossary.
italic	Italic type indicates book titles, emphasis, or placeholder variables for which you supply particular values.
`monospace`	Monospace type indicates commands within a paragraph, URLs, code in examples, text that appears on the screen, or text that you enter.

1

About Data Transformation Platform

Data Transformation Pipelines enable organizations to quickly respond to changes in regulations, adapt to new product introduction, counter more sophisticated money laundering techniques in a quick and cost-effective manner by removing dependency on Oracle to develop the scenarios and extensions most appropriate to their business requirements.

Purpose

The purpose of the Data Transformation Platform is to provide Financial Institutions with full scenario configurability and extensibility, allowing them to quickly create scenario pipelines using complex logic, advanced configuration, leveraging extended data sources.

User Personas

This guide is intended for users who are responsible for implementing the scenarios in line with the Risk analysis formulated by the compliance team and by Compliance Officers who need to understand the additional possibilities offered by scenarios based on data they are capturing, in order to make their AML program more effective and efficient. Oracle expects these users to be senior compliance officers, who have significant understanding of their data, the specific behaviors of interest, and the logic required to generate meaningful alerts.

2

Getting Started with Data Transformation Pipelines

This section explains how to create, copy, and configure data transformation pipelines, and how to use the widgets available in the DTP Pipeline Designer.

To access the DTP Pipeline Designer, follow these steps:

1. In the Applications landing page, click the **Navigation Menu** to access the Navigation List. The Navigation List displays the list of modules.
2. Expand **Data Transformation Pipeline** in the Navigation List to view all DTP menu options.
   - DTP Pipeline Designer
   - DTP Threshold Manager
   - DTP Jobs
   - DTP Audit History
   - Data Transformation Pipeline Admin

These features are described in this guide.

Downloading DTP Pipelines

The Download Pipelines feature enables you to download a version of an existing pipeline. The pipelines are saved in JSON format which can be imported into the application.

This is frequently used to download a pipeline from a non-production environment and then import the pipeline into a production environment.

1. Navigate to the DTP Pipeline Designer page.
2. Click the **Action** button corresponding to the pipeline that you want to download.
3. Click **Download** .
4. Select **JSON**. The Download Pipeline pane displays.
5. Enter the version of the pipeline to download in the **Pipeline Version** field. You can only enter numerals here.

> **Note**
>
> This should be the same as the current version of the pipeline you are downloading.

6. Click **Download**.
7. Select a folder to save the pipeline.

8. Enter a file name, using the following naming convention: fileName . filenameformat : pipelineName_versionNumber.json
9. Click **Save**. The pipeline JSON is downloaded.

Importing DTP Pipelines

You can import a new JSON version of an existing Data Transformation pipeline. This is frequently done to import a pipeline from a non-production environment into a production environment.

Before importing a pipeline, you must first download the pipeline using the steps in Downloading DTP Pipelines.

> **Note**
>
> You cannot import a pipeline that already exists in the environment. Before importing a pipeline, verify that this pipeline does not already exist in the environment. If so, delete the pipeline before importing the new version.

To import a pipeline, follow these steps:

1. Navigate to the DTP Pipeline Designer page.
2. Click **Import**. The Import pane opens.
3. Enter the required version of the pipeline to import in the **Version** field. You can also use the arrows to select the version number.

   The Version number provided should be the same as that in the downloaded .JSON file.
4. Drag and Drop the .JSON file of the pipeline to import into the box. You can also click the **Drag and Drop** box to browse to the location of the .JSON file. The pipeline should have the following naming convention: fileName . filenameformat : pipelineName_versionNumber.json

   > **Note**
   >
   > You can only import the .JSON file of a DTP pipeline.
5. Click **Import**. The pipeline is imported and added to the My Pipelines list.

Copying DTP Pipelines

You can copy an existing pre-configured or user-configured pipeline and then configure it to meet your specific business requirements. This reduces the effort of creating and configuring a new pipeline when you only need to modify a few components of the pipeline.

Pre-configured pipelines that are included out-of the box cannot be modified. In order to customize these pipelines, you must first create a copy of those pipelines and save it as a new pipeline. You can then configure the newly copied pipeline according to your requirements.
To copy a pipeline, follow these steps.

1. Navigate to the DTP Pipeline Designer page.
2. Click the **Action** button ••• corresponding to the pipeline that you want to copy.

3. Click **Copy**  . The Copy Pipeline pane displays.
4. Provide the details as described in the following table.

Table 2-1 Fields for Copying Pipelines and their Descriptions

Field	Description
Name	Displays the name of the pipeline that you want to copy. You cannot modify this field.
Copy As	Enter the name for the new pipeline that you want to create by copying the existing pipeline. This field allows only the following characters: A-Z a-z 0-9 '-' For example, Abc-123 or Abc123.
Pipeline Focus	Displays the focus for the pipeline that you want to copy. This will be the focus for the new pipeline. You cannot modify this field.
Search Tags	Enter the keywords for the pipeline. These keywords can be used as search tags while searching for a pipeline. Search tags are also used to group pipelines of the same type. These search tags appear as filters in the dashboard.
Description	Enter the description for the pipeline.

5. Click **Save**. The pipeline is copied with all widgets and values provided in the original pipeline.

   You can now modify these components, if desired. See Editing Pipelines for more information.

> **Note**
>
> When editing a pipeline, consider the impact your changes may have on subsequent widgets in the pipeline.
> For example, removing a mapped table or column in the Data Forge may impact how an existing Aggregator functions.

Adding DTP Pipelines

You can add a new pipeline.

To add a new pipeline, follow these steps:

1. Navigate to the DTP Pipeline Designer page.
2. Click **Add Pipeline**. The Add Pipeline pane opens.
3. Provide the details as described in the following table.

Table 2-2 Fields for Creating Pipelines and their Descriptions

Field	Description
Name	Enter the name for the pipeline. This field allows only the following characters: A-Z a-z 0-9 '-' For example, Abc-123 or Abc123.

Table 2-2 (Cont.) Fields for Creating Pipelines and their Descriptions

Field	Description
Pipeline Type	Select **Data Transformation** from the **Pipeline Type** drop-down list.
Focus	Select the **Focus** for your pipeline from the drop-down list. For more information about Focuses, see Creating Focus Types.
Add Search Tags	Enter the keywords for the pipeline. These keywords can be used as search tags while searching for a pipeline. Search tags are also used to group pipelines of the same type. These search tags appear as filters in the Pipeline page.
Description	Enter the description for the pipeline.

4. Click **Create**. A new pipeline is created and displayed in the Pipeline page. You can perform the required configurations in the newly created pipeline using the Hamburger menu or by right-clicking on the canvas to view available widgets.

   For more information about widgets available in DTP pipelines, see Widgets in Data Transformation Pipelines.

5. Click **Save** to save the pipeline.

Editing DTP Pipelines

You can configure user-defined pipelines to better address your business needs as requirements change.

> **Note**
>
> Pre-configured pipelines cannot be edited. You can only edit user-defined pipelines.

To edit pipelines, follow these steps:

1. Navigate to the DTP Pipeline Designer page.
2. Click the pipeline name of the pipeline you want to edit. The DTP Pipeline Designer displays the widgets used in this pipeline.
3. Perform the required configurations using the Hamburger menu or by right-clicking on the canvas to view available widgets.

   For more information about widgets available in DTP pipelines, see Widgets in Data Transformation Pipelines.

   > **Note**
   >
   > When editing a pipeline, consider the impact your changes may have on subsequent widgets in the pipeline. For example, removing a mapped table or column in the Data Forge may impact how an existing Aggregator functions.

4. Click **Save** to save the pipeline.

Editing Pipeline Descriptions

You can edit the description and search tags for user-created pipelines.

> **Note**
>
> Pre-configured pipelines cannot be edited. You can only edit user-configure pipelines

To edit the description and search tags, follow these steps:

1. Navigate to the DTP Pipeline Designer page.
2. Click the Action button corresponding to the pipeline that you want to copy.
3. Click **Edit** . The Edit Pipeline pane displays.
4. Modify the required details.

Table 2-3 Fields for Editing Pipelines and their Descriptions

Field	Description
Name	Displays the name of the pipeline that you want to edit. You cannot modify this field.
Pipeline Focus	Displays the focus for the pipeline that you want to edit. This will be the focus for the new pipeline. You cannot modify this field.
Search Tags	Add or remove keywords for the pipeline. These keywords can be used as search tags while searching for a pipeline. Search tags are also used to group pipelines of the same type. These search tags appear as filters in the dashboard.
Description	Edit the description for the pipeline.

5. Click **Save**.

Deleting DTP Pipelines

You can delete a pipeline, if required for your implementation.

To delete a pipeline, follow these steps:

1. Navigate to the DTP Pipeline Designer page.
2. Click the **Action** button corresponding to the pipeline that you want to delete.
3. Click **Delete** corresponding to the pipeline that you want to delete. A dialog box opens.
4. Enter a comment explaining why you are deleting the pipeline.
5. Click **Delete**. The selected pipeline is deleted.

Widgets in Data Transformation Pipelines

Depending on the pipeline type, specific widgets are available in the widgets pane of the pipeline. This topic describes the widgets in Data Transformation pipelines.

The following table describes the widgets available in Data Transformation pipelines.

Table 2-4 Data Transformation Pipeline – Widgets and Descriptions

Widget	Name	Description
	Calendar	Use this widget to add a Calendar, which allows you to designate and configure the schedule the scenario pipeline should be run according to.
	Data Forge	Use this widget to select the source(s) (Tables) which form the dataset to be further used in the pipeline.
	Business Filter	Use this widget to filter the data in the pipeline to create a subset of the data records which are available by applying the business filters. This allows you to search and analyze behaviors of interest.
	Data Transformation	Use this widget to write complex expressions and transformation in Python.
	Aggregator	Use this widget to define aggregators using available aggregations in the widget.
	Evaluation	Use this widget to configure an evaluation. Evaluations are used to define conditions for the aggregations that are defined in the aggregator. Relevant data (aggregators) is compared against these conditions using logical comparisons to identify the early signals of increasing risk exposures in various areas of an enterprise, resulting in event generation.
	Create Event	Use this widget to produce an event. This widget accepts inputs from evaluation and generates events based on evaluated output data from the evaluation widget.

Using the Calendar Widget

The Calendar widget allows you to designate and configure the calendar for the scenario's look back and frequency periods.

Before adding or configuring a calendar, you must first have created the pipeline.

1. In the Pipeline Designer page, select the pipeline you want to add or modify the calendar for. The Pipeline Canvas displays.
2. Right-click on the Canvas to display the list of widgets. Select **Calendar**  . The Calendar widget is added to the pipeline Canvas. You can drag and drop the widget within the Canvas as needed.
3. Click **Options**  then click **Edit**  . The Calendar pane displays.

4. Provide the details as described in the following table.

Table 2-5 Fields for Creating Calendars and their Descriptions

Field	Description
Name	Enter the name for the calendar. This field allows only the following characters: A-Z a-z 0-9 '_' For example, Abc_123 or Abc 123
Lookback Type	Select the Lookback Type used in this pipeline using the drop-down list. This is the range of days of data monitored in each run of the detection process. Options include Days or Months. This controls the unit in which both the lookback period and frequency period are expressed.
Days	Select either **Calendar Days** or **Business Days** from the drop-down list.
Weekdays	Select all the days which should be considered Week Days from the list.
Holiday Master	Select the applicable holidays which should be considered in this calendar. This section is populated by the data provided in the FCC_AM_HOLIDAY_MASTER table. For more about how this table is populated, see AMLHolidayMasterDataLoad Batch Details.

5. Click **Save**. A confirmation message displays.

Using the Data Forge Widget

The Data Forge widget allows you to select the source(s) (Tables) which form the dataset to be further used in the pipeline.

Before adding or configuring a Data Forge, you must first have created the pipeline.

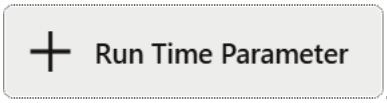
1. In the Pipeline Designer page, select the pipeline where you want to add or modify the Data Forge. The Pipeline Canvas displays.
2. To add a Data Forge to a Calendar, click **Add To**  , and then select **Data Forge**  .
   - Alternately, you can right-click on the Canvas to display the list of widgets, select **Data Forge** and associate the Data Forge with the Calendar.
   - You can also use the hamburger menu on the left side to display the Node Picker, which lists any externalized data forges already created. Drag and drop the data forge to the pipeline as desired.
3. Click **Options**  then click **Edit**  . The Data Forge pane displays.
4. Under **Basic Configuration**, provide the following details:
   a. Enter the **Name** for the Data Forge. This field allows only the following characters: A-Z a-z 0-9 '_' For example, Abc_123 or Abc 123
   b. Select the **Type** from the drop-down list.
   c. The **Externalize** selector allows you to externalize this data forge so that it can be used in additional pipelines. To access all externalized data forges, click the

hamburger menu on the left side to display the Node Picker, which lists any externalized data forges already created.

d. Select one or more **Source Tables** using the drop-down list. Use the search bar in the list to find specific tables easily.

   To remove a source table from the Data Forge, click **Remove** ✕ beside the table name in the Source Tables field.

e. The **Connected Data Forges** lists any data forges which are associated with this data forge.

f. You can add a Runtime Parameter to the Data Forge by clicking **Add Run Time Parameter** 

. For more information about how to create run time parameters, see Creating Runtime Parameters.

5. The **Advanced Configuration** tab displays any Joins and Filters associated with this Data Forge.

> **Note**
>
> At least two sources (tables/dataforges) must be selected or connected for the join section to be enabled in the **Advance Configuration** tab.

- Expand **Joins** to view or create Joins associated with this Data Forge. The Output box provides a summary of all current Joins, with the Join configurations listed in detail below. You can add new Joins using the steps in Creating Joins.
- Expand **Filters** to view or create Filters associated with this Data Forge. The Output box provides a summary of all current Filters, with the Filter configurations listed in detail below. You can add new Filters using the steps in Creating Filters .

6. Use the **Mapped Columns** tab to view and map columns within the Data Forge. To map additional columns, follow these steps:

a. In the Mapped Columns tab, select a Source Table/Dataforge from the drop-down list. The available columns in this table display.

b. Select a Column from the list. Use the search bar in the list to find specific tables easily.

c. Click **Map**  to map the column. The column displays in the Mapped Columns list. You can edit the Mapped Column Name, if desired.

d. Alternately, click **Add Expression**  to open the Expression Builder. The Expression Builder is used to define free flow text filter conditions. To use the Expression Builder, follow these steps:

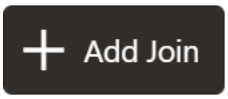
a. Select the required Dataset, Attribute and Runtime Parameters and operators. The resulting condition is displayed in the Condition field.

b. Click **Save**  to save the changes.

7. When you have finished configuring your Data Forge, click **Save**.

Creating Joins

Joins allow you to combine or group multiple tables using various join operators.

1. In the Advanced Configuration tab of the Data Forge pane, click **Add Join** 

. A new Join row is added to the list.
2. Use the **Include** selector to indicate whether or not this Join should be included in this Data Forge.
3. Select a **Join Operator** from the drop-down list.

   The following types of join operators are available:

   - **Inner Join:** The Inner Join selects all rows from both participating tables as long as there is a match between the columns.
   - **Left Join:** The Left Join returns all rows from the left table, with the matching rows in the right table.
   - **Right Join:** The Right Join returns all rows from the right table, with the matching rows in the left table.
   - **Full Join:** The Full Join combines the results of both the left and right outer joins and returns all rows from the tables on both sides
4. Add a Join condition to the Join table by following these steps. You can add multiple groups and multiple conditions under each group.

   a. Click **Add Group** and then **Add Condition**  .

   b. Specify rules for the condition. You can define the conditions using one of the following:

      - **Expression Builder:** You can form conditions using all the operators given in the Expression Builder. The Expression Builder is used to define free flow text filter conditions. To define a filter condition using the Expression Builder, follow these steps:

        i. Click **Exp**. The Expression Builder dialog box is displayed.

        ii. Select the required Dataset, Attribute and Runtime Parameters and operators. The resulting condition is displayed in the Condition field.

        iii. Click **Save**  to save the changes.

      - **Tables:** You can define conditions using the various columns of tables. The columns of the two tables are compared with each other using the required operators. To define conditions using tables, follow these steps:

        i. Select a Table or Data Forge for the left-hand side from the drop-down list.

ii. Select a Column for the left-hand side from the drop-down list.

iii. Select an Operator from the drop-down list.

iv. Select a Table or Data Forge for the right-hand side from the drop-down list.

v. Select a Column for the right-hand side from the drop-down list.

- **Text:** You can define filter conditions using text. A particular column in a table is compared with the input text using the required operators. To define filter conditions using text, follow these steps:

  i. Click **Text**.

  ii. Select a Table or Data Forge for the left-hand side from the drop-down list.

  iii. Select a Column for the left-hand side from the drop-down list.

  iv. Select an Operator from the drop-down list.

  v. Enter the text in the field on the right-hand side.

Creating Filters

You can configure a filter by defining various filter conditions.

1. In the Advanced Configuration tab of the Data Forge pane, click **Add Group** 

. A new Filter Group row is added to the list.
2. Use the **Include** selector to indicate whether or not this Filter should be included in this Data Forge.
3. Add a Filter condition by following these steps. You can add multiple groups and multiple conditions under each group.

   a. Click **Add Condition** 
.

   b. Specify rules for the condition. You can define the conditions using one of the following:

   - **Expression Builder:** You can form conditions using all the operators given in the Expression Builder. The Expression Builder is used to define free flow text filter conditions. To define a filter condition using the Expression Builder, follow these steps:

     i. Click **Exp**. The Expression Builder dialog box is displayed.

     ii. Select the required Dataset, Attribute and Runtime Parameters and operators. The resulting condition is displayed in the Condition field.

     iii. Click **Save** 
to save the changes.

   - **Tables:** You can define conditions using the various columns of tables. The columns of the two tables are compared with each other using the required operators. To define conditions using tables, follow these steps:

     i. Select a Table or Data Forge for the left-hand side from the drop-down list.

     ii. Select a Column for the left-hand side from the drop-down list.

iii. Select an Operator from the drop-down list.

iv. Select a Table or Data Forge for the right-hand side from the drop-down list.

v. Select a Column for the right-hand side from the drop-down list.

- **Text:** You can define filter conditions using text. A particular column in a table is compared with the input text using the required operators. To define filter conditions using text, follow these steps:

  i. Click **Text**.

  ii. Select a Table or Data Forge for the left-hand side from the drop-down list.

  iii. Select a Column for the left-hand side from the drop-down list.

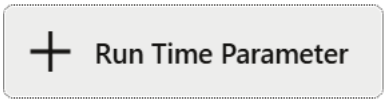
  iv. Select an Operator from the drop-down list.

  v. Enter the text in the field on the right-hand side.

Creating Runtime Parameters

A Runtime parameter is a variable whose value can be defined and then called from within that same pipeline.

When you define a runtime parameter, you enter the default value to use. When you create or edit a job that includes a pipeline with runtime parameters, you can specify another value to override the default.
To create a runtime parameter, follow these steps:

1. Click **Add Runtime Parameter** 

. The New Runtime Parameter dialog box displays.
2. Provide the details as described in the following table:

Table 2-6 Fields in New Runtime Parameter and their Descriptions

Field	Description
Name	Enter the name for the runtime parameter. Any spaces ( ) given in the Name field will be replaced with underscore (_) while saving the runtime parameter.
Description	Enter the description for the runtime parameter.
Datatype	Select the datatype for the runtime parameter from the drop-down list.
Default Value	Provide the default values for the runtime parameter.

3. Click **OK**.The runtime parameter is created.

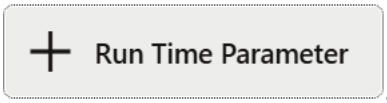
Using the Business Filter Widget

The Business Filter widget to filter the data in the pipeline to create a subset of the data records which are available by applying the business filters. This allows you to search and analyze behaviors of interest

Before adding or configuring a Business Filter, you must first have created the pipeline and added either a Data Forge or Data Transformation.

1. In the Pipeline Designer page, select the pipeline where you want to add or modify the Business Filter. The Pipeline Canvas displays.
2. To add a Business Filter to a Data Forge, click **Add To**  , and then select **Business Filter**  . Alternately, you can right-click on the Canvas to display the list of widgets, select **Business Filter** and associate the Business Filter with the Data Forge.

   > **Note**
   >
   > Business Filters must be associated with a Data Forge.

3. Click **Options** then click **Edit**  . The Business Filter pane displays.
4. Under **Basic Configuration**, provide the following details:
   a. Enter the **Name** for the business filter. This field allows only the following characters: A-Z a-z 0-9 '_' For example, Abc_123 or Abc 123
   b. **Connected Sources** lists the name of the connected Data Forge or Data Transformation.
5. Click **List of Filters** to add a Business Filter which has already been created for columns in your data set. Use the selectors to select one or more filters as needed.
6. The **Questionnaire** tab lists the mapped filters, which are the filters you selected in the previous step. Based on the Filter Type, you can further refine the filter.
   a. For Include-Exclude types, use **Include** and **Exclude** to include or exclude one or more specific fields. For example, Include Retail Banking.
   b. For Yes-No types, use **Yes** and **No** to choose whether to enable this filter.
   c. For Filter types, use the drop-down lists to select the operators and code values. Select **Parameter** if you want to filter based on one or more specific fields.
   d. Use **Add Filter** to add a new business filter to the List of Filters.. For more information about how to add a filter, see Adding New Business Filters
   e. You can add additional conditions.
   f. You can add a Runtime Parameter to the Business Filter by clicking **Add Run Time Parameter** 

. For more information about how to create run time parameters, see Creating Runtime Parameters.
7. Click **Save**. A confirmation message displays.

Adding New Business Filters

You can create new business filters as needed to meet your business requirements.

1. In the Business Filter pane, click **Add Filter**.
2. Provide the details as described in the following table.

Table 2-7 Fields for Creating Business Filters and their Descriptions

Field	Description
Name	Enter the **Name** for the business filter. This field allows only the following characters: A-Z a-z 0-9 '_' For example, Abc_123 or Abc 123
Entities	Select which data set to create the filter from using the **Entities** drop-down list.
Attribute	Select the column on which you will be filtering the data from the **Attribute** drop-down list. All columns that were mapped in the Data Forge or Columns resulting from the connected Data Transformation widget will be displayed.
Filter Type	Select from the following options: • **Include-Exclude**: Use the Input Codes section to select the input which should be included or excluded. Use the Input Codes section to feed the inputs in the form of CODES and logical Names. (for example, IND-Individual). • **Yes-No** • **Filter**

3. Add the filter, using one of the following options. A confirmation message displays.
   - Click **Add** to add this Business Filter to the List of Filters.
   - Click **Add & Attach** to add this Business Filter to the List of Filters and attach it to the current data set.
4. When you have finished creating filter, click **Save**.

> **Note**
>
> Oracle recommends creating the following commonly used Business Filters in order to simplify future pipeline configuration.

Table 2-8 Common Business Filters

Business Filter	Purpose
Account Business Type	Indicates which types of functional areas in which the account does business should be included when detecting behaviors of interest.
Account Holder Type	Indicates which types of internal or external customers should be included when detecting behaviors of interest.
Transaction Type	Indicates which transaction types are included when detecting behaviors of interest.
Transaction Purpose	Indicates which transaction purposes should be included when detecting behaviors of interest.
Party Role	Indicates which party roles are included when detecting behaviors of interest.

Creating Custom Filters

The Custom Filter allows you to create business filters using the Expression Builder.

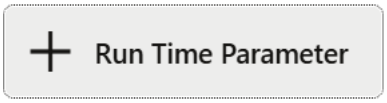
You can form filter conditions using all the operators given in the Expression Builder. The Expression Builder is used to define free flow text filter conditions. To define a filter condition using the Expression Builder, follow these steps:

1. In the Business Filter pane, click **Custom Filter**.
2. Select the required Dataset, Attribute, and operators to create the filter. The resulting condition is displayed in the **Condition Value** box.
3. Click **Save**  to create this filter
4. Create additional filters, as desired. When you have finished creating all the filters needed for this pipeline, click **Save** to save the changes.

Creating Runtime Parameters

A Runtime parameter is a variable whose value can be defined and then called from within that same pipeline.

When you define a runtime parameter, you enter the default value to use. When you create or edit a job that includes a pipeline with runtime parameters, you can specify another value to override the default.
To create a runtime parameter, follow these steps:

1. Click **Add Runtime Parameter** 

. The New Runtime Parameter dialog box displays.
2. Provide the details as described in the following table:

Table 2-9 Fields in New Runtime Parameter and their Descriptions

Field	Description
Name	Enter the name for the runtime parameter. Any spaces ( ) given in the Name field will be replaced with underscore (_) while saving the runtime parameter.
Description	Enter the description for the runtime parameter.
Datatype	Select the datatype for the runtime parameter from the drop-down list.
Default Value	Provide the default values for the runtime parameter.

3. Click **OK**.The runtime parameter is created.

Using the Data Transformation Widget

The Data Transformation widget allows you to write and import complex expression and transformation in Python and Java.

To use this widget effectively, you must have a working knowledge of scripting and Java or Python.

1. Create a pipeline, as shown in Adding DTP Pipelines.
2. Add a Data Transformation widget to the pipeline by right-clicking the Canvas to display the list of widgets and selecting **Data Transformation**  .

   The Data Transformation widget can be stand-alone or associated with another widget, such as Business Filter, Evaluation, or Aggregator.

3. Click **Options** then click **Edit**  . The Data Transformation pane displays.
4. In Basic Configuration, provide the following details:
   a. Enter a **Name** for this Data Transformation.
   b. Select a **Runtime Parameter** from the drop-down list. To add a new Runtime Parameter, see Creating Runtime Parameters.
   c. For Data Transformations associated with another widget, map the columns in **Column Definition** using **Add Columns**. The **View Input Table Columns** shows which columns were mapped in the connected Data Forge.
5. The **Define Transformation** tab displays the script options.
   - **Standard**
     a. Select the **Type** from the drop-down list.
     b. Select the **Transform** from the drop-down list.
     c. Select **Interpreters** from the drop-down list. Python allows you call a custom python script.
        i. In the **Script** section, provide a description.
        ii. Enter or paste the python script you want to execute.

        > **Note**
        >
        > If this is a script you intend to reuse, you can create a template by clicking **Templates**, then **Add**. Enter the pertinent details about the script, enter or paste the script into the box, and click **Save**. Existing templates can be modified by clicking **Edit**. Editing a template updates the script in every pipeline which uses this template.

        To use an existing template, select **Use Template** and select the desired template from the **Script** drop-down list.

        iii. Click **Save**.
   - Selecting **Custom** allows you to create a notebook.

Notebooks allow you to add and run paragraphs, invalidate session, edit, add, export the notebook, and so on. To understand the icons which display in the Notebook, see Common Screen Elements.

a. Click **Add Python Paragraph** to add a paragraph in the notebook and fetch the runtime parameters. This is available for all the interpreters.

b. Add paragraphs to the Notebook as desired.

c. Click **Play** to run the script and display the label.

6. You can import an existing notebook by selecting **Import Notebook**.
   - Click **Import Archive File** to select the .dsnb file from the file selector dialog and select a file. Click **Open**.
   - OR
   - Drag and drop the file into the **Import Archive File** field.

Common Screen Elements in the Notebook

This topic describes common screen elements in the notebook that can be used to perform quick actions when preparing and executing the notebooks.

Table 2-10 Elements in the Notebook

Icon Name	Action/Description
Modify Notebook 	Modifies the details of a notebook, such as a name, description, and (or) tags.
Hide Code 	Hides or shows the Code Section in all the paragraphs in a notebook.
Hide Result 	Hides or shows the Results Section in all the paragraphs in a notebook.
Re-initialize session 	Re-initialize the paragraph in a notebook. **Note**: By default, the Start Widget gets executed automatically on clicking the Re-initialize icon. All other paragraphs need to be executed manually.

Table 2-10 (Cont.) Elements in the Notebook

Icon Name	Action/Description
Read-Only 	Sets the notebook to read-only mode. **Note**: The notebook is protected from edit, clear result, delete, share, reset session, and run paragraphs in Read-only mode.
Write 	Set the notebook to write mode.
Run Paragraphs 	Executes all the paragraphs in a notebook in sequential order.
Invalidate Session 	Resets any connection or code executed in a notebook.
Delete Notebook 	To delete the selected notebook.
Clear Result 	Clears results for all the paragraphs in a notebook. **Note**: This action clears all the results. You must rerun the paragraphs to view the results.
Clear Paragraph Dependencies 	Remove all defined paragraph dependencies.

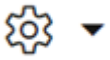
Table 2-10 (Cont.) Elements in the Notebook

Icon Name	Action/Description
Open as an iframe 	Opens a notebook in an Iframe. This allows a notebook to be embedded inside another webpage.
Share Notebook 	Share the notebook with another user, user group, or role.
Clone Notebook 	Creates a copy of a notebook. All paragraphs in the current notebook are replicated in the new notebook. The cloned notebook is created with the default name, Copy of <Current Notebook Name>.
Export Notebook 	Export the notebook to your computer as a DNSB file.
Print Notebook 	To print a notebook in the PDF format and save it in your local machine.
Default Template 	To apply the overall look and feel of the notebook using the default template.
Layout 	Sets the preferred layout. The available layouts are Zeppelin and Jupyter.

Table 2-10 (Cont.) Elements in the Notebook

Icon Name	Action/Description
Default Template 	Applies the overall look and feel of the notebook using the default template.
Show Panel 	Shows or hides the Paragraph Settings Bar Commands, Results Toolbar, and Settings Dialog for a selected paragraph in a panel to the notebook's right.
Attach Credentials 	You can use this option to attach credentials (wallet and password) to the notebook to enable secure data access.
Versioning 	You can use this option to create versions for your notebook, which helps you analyze the changes based on the version control.
Run Paragraph 	To execute the code or query in a paragraph.
Enter Dependency Mode 	To add or remove dependent paragraphs. Paragraphs with dependent paragraphs are executed in the dependency order.
Comments 	To add comments to a paragraph.

Table 2-10 (Cont.) Elements in the Notebook

Icon Name	Action/Description
Expand 	To expand a paragraph and view the paragraph in full-screen mode.
Show/Hide Line Numbers 	To show or hide line numbers in the code in a paragraph. **Note**:This button is applicable only to the code section.
Visibility 	To manage the visibility settings in a paragraph. It controls how a paragraph may be viewed by the author and other users who have access to the notebook.
Settings 	Users can perform the following actions: • Resize the width of a paragraph • Change the order of placement of the paragraphs by moving them up or down • Run the all above or all below paragraph from the selected paragraph. • Clear the paragraph result • Open the notebook in Embedded window • Disable the run action • Clone the paragraph • Delete a paragraph

Using the Aggregator Widget

Relevant data is compared against aggregators to identify the early signals of increasing risk exposures in various areas of an enterprise.

Before adding or configuring an Aggregator, you must first have created the pipeline, data forge, and business filter.

1. In the Pipeline Designer page, select the pipeline where you want to add or modify the Aggregator. The Pipeline Canvas displays.
2. To add an Aggregator to a Business Filter, click **Add To**  , and then select **Aggregator**.

    Alternately, you can right-click on the Canvas to display the list of widgets, select **Aggregator** and associate the Aggregator with the Business Filter.

   An aggregator can be associated to another aggregator, if desired, to further refine the aggregated data. The fields will be populated only by data provided in the parent aggregator. When two aggregators are linked to another aggregator a new tab,

Aggregator Joins , displays. For information on how to create aggregator joins, see Creating Aggregator Joins.

3. Click **Options** then click **Edit**  . The Aggregator pane displays.
4. Under **Basic Configuration**, enter the **Name** for the aggregator. This field allows only the following characters: A-Z a-z 0-9 '_' For example, Abc_123 or Abc 123s:
5. Click **List of Aggregators** to add an Aggregator which has already been created in your data set. Use the selectors to select one or more filters as needed. Use **View** to view the Aggregator Details.
6. The **Aggregators** tab lists the mapped aggregators, which are the aggregators you selected in the previous step. Click **Edit** to further define the aggregator.
   - Clicking **Edit** on a standard aggregator displays the add/edit aggregator tab, and the aggregator can be edited.
   - Clicking **Edit** on a custom aggregator opens the **Custom Aggregator**, which allows you to form aggregator conditions using all the operators given in the Expression Builder. The Expression Builder is used to define free flow text conditions. To define a condition using the Expression Builder, see Creating Custom Aggregators.
7. Click **Save**. A confirmation message displays.

Adding and Editing Aggregators

You can create new aggregators or edit existing aggregators as needed to meet your business requirements.

1. Select the **Aggregator** from the drop-down list. This list contains the types of aggregation which are available for this Business Filter, such as Sum, Count or Average Monthly.
2. Enter a **Group By** clause. This allows you to group aggregators by the specified attribute across the pipeline. Grouping by a certain attribute will be applied to all aggregators used in the pipeline.
3. Select the **Attribute** from the drop-down list. This list contains the data forge tables and mapped columns.
4. Select the required **Look Back** option and provide the required Value details. You can further configure the Look Back period for the aggregator when building the expression.
5. Select the **Filter Attribute**.
6. Select the **Filter Operator**.
7. Select the required **Filter Value** option and provide the required details.
8. Add the aggregator using one of the following options. A confirmation message displays.
   - Click **Add** to add this aggregator to the List of Aggregators.
   - Click **Add & Attach** to add this aggregator to the List of Aggregators and attach it to the current data set.
9. When you have finished creating aggregators, click **Save**.

Creating Custom Aggregators

The Custom Aggregator option allows you to create aggregators using the Expression Builder.

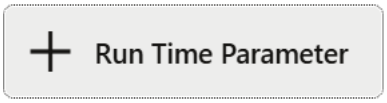
You can form aggregator conditions using all the operators given in the Expression Builder. The Expression Builder is used to define free flow text aggregator conditions. To define an aggregator condition using the Expression Builder, follow these steps:

1. In the Aggregator pane, click **Custom Aggregator**.
2. Enter the **Name** for the Aggregator. This field allows only the following characters: A-Z a-z 0-9 '_' For example, Abc_123 or Abc 123.
3. Select the required Attributes and Operators to create the Aggregator. The resulting condition is displayed in the **Condition Value** box.
4. Select the required **Look Back** option and provide the required Value details. You can further configure the Look Back period for the aggregator when building the expression.
5. Add the aggregator using one of the following options. A confirmation message displays.
   - Click **Add** to add this aggregator to the List of Aggregators.
   - Click **Add & Attach** to add this aggregator to the List of Aggregators and attach it to the current data set.
6. When you have finished creating aggregators, click **Save**.

Creating Runtime Parameters

A Runtime parameter is a variable whose value can be defined and then called from within that same pipeline.

When you define a runtime parameter, you enter the default value to use. When you create or edit a job that includes a pipeline with runtime parameters, you can specify another value to override the default.
To create a runtime parameter, follow these steps:

1. Click **Add Runtime Parameter** 

. The New Runtime Parameter dialog box displays.
2. Provide the details as described in the following table:

Table 2-11 Fields in New Runtime Parameter and their Descriptions

Field	Description
Name	Enter the name for the runtime parameter. Any spaces ( ) given in the Name field will be replaced with underscore (_) while saving the runtime parameter.
Description	Enter the description for the runtime parameter.
Datatype	Select the datatype for the runtime parameter from the drop-down list.
Default Value	Provide the default values for the runtime parameter.

3. Click **OK**.The runtime parameter is created.

Creating Aggregator Joins

When two aggregators are linked to another aggregator you should create Aggregator Joins.

This tab will only display when two parent aggregators are associated with a child aggregator.

1. In the Aggregator Joins tab of the Aggregator pane, click **Add Join** 

. A new Join row is added to the list.
2. Use the **Include** selector to indicate whether or not this Join should be included in this aggregator.
3. Select a **Join Operator** from the drop-down list.

   The following types of join operators are available:

   - **Inner Join:** The Inner Join selects all rows from both participating tables as long as there is a match between the columns.
   - **Left Join:** The Left Join returns all rows from the left table, with the matching rows in the right table.
   - **Right Join:** The Right Join returns all rows from the right table, with the matching rows in the left table.
   - **Full Join:** The Full Join combines the results of both the left and right outer joins and returns all rows from the tables on both sides
4. Add a Join condition to the Join table by following these steps. You can add multiple groups and multiple conditions under each group.

   a. Click **Add Group** and then **Add Condition**  .

   b. Specify rules for the condition. You can define the conditions using one of the following:

      - **Expression Builder:** You can form conditions using all the operators given in the Expression Builder. The Expression Builder is used to define free flow text filter conditions. To define a filter condition using the Expression Builder, follow these steps:

        i. Click **Exp**. The Expression Builder dialog box is displayed.

        ii. Select the required Dataset, Attribute and Runtime Parameters and operators. The resulting condition is displayed in the Condition field.

        iii. Click **Save**  to save the changes.
      - **Tables:** You can define conditions using the various columns of tables. The columns of the two tables are compared with each other using the required operators. To define conditions using tables, follow these steps:

        i. Select a Table or Data Forge for the left-hand side from the drop-down list.

        ii. Select a Column for the left-hand side from the drop-down list.

        iii. Select an Operator from the drop-down list.

iv. Select a Table or Data Forge for the right-hand side from the drop-down list.

v. Select a Column for the right-hand side from the drop-down list.

- **Text:** You can define filter conditions using text. A particular column in a table is compared with the input text using the required operators. To define filter conditions using text, follow these steps:

  i. Click **Text**.

  ii. Select a Table or Data Forge for the left-hand side from the drop-down list.

  iii. Select a Column for the left-hand side from the drop-down list.

  iv. Select an Operator from the drop-down list.

  v. Enter the text in the field on the right-hand side.

Using the Evaluation Widget

Evaluations are used to define conditions for the measures that are defined in the aggregator. Evaluations perform logical comparisons against these conditions to generate events.

Before adding or configuring an Evaluation, you must first have created the pipeline, data forge, business filter, and aggregator.

1. In the Pipeline Designer page, select the pipeline where you want to add or modify the Evaluation. The Pipeline Canvas displays.
2. To add an Evaluation to an Aggregator, click **Add To**  , and then select **Evaluation**  . Alternately, you can right-click on the Canvas to display the list of widgets, select **Evaluation** and associate the Evaluation with the Aggregator.
3. Click **Options**  then click **Edit**  . The Evaluation pane displays.
4. Under **Basic Configuration**, enter the **Name** for the Evaluation. This field allows only the following characters: A-Z a-z 0-9 '_' For example, Abc_123 or Abc 123s:
5. The **Advanced Configuration** tab displays the detection logic and event generation criteria. Event generation criteria are shown in formula-style expressions in the Output box.
6. You can add multiple groups and multiple conditions under each group. Add conditions to the Group by clicking **Add Group** and then **Add Condition** 
7. Select the required Aggregator, operators, and conditions to build the evaluation. You can define the evaluation conditions using one of the following:

   - Expression Builder: You can form filter conditions using all the operators given in the Expression Builder. The Expression Builder is used to define free flow text conditions. To define a condition using the Expression Builder, follow these steps:

     a. Click **Exp**. The Expression Builder dialog box is displayed.

     b. Select the required Attribute and Runtime Parameters and operators. The resulting condition is displayed in the Condition Value field.

     c. Click **Save** to save the changes.

- Text: You can define filter conditions using text. A particular column in a table is compared with the input text using the required operators. To define filter conditions using text, follow these steps:
  a. Click **Text**.
  b. Select an aggregator and operator, and then enter the text in the field on the right-hand side.
  c. Click **Save** to save the changes.

8. Use the Include selector to choose whether to include the condition group in this Evaluation.
9. Click **Save**. A confirmation message displays.

Using the Create Event Widget

This widget accepts inputs from evaluation and generates events based on evaluated output data from the evaluation widget

In Scenario Pipelines, the Create Event widget is the final part of the pipeline and is used to produce an event. An event is a record of one or more pattern matches in a detection run, which is a signal for further investigation.

The Create Event widget can only be attached to an Evaluation widget. Multiple Evaluation widgets can be connected to one Create Event widget.
To create an event, follow these steps:

1. In the Pipeline Designer page, select the pipeline where you want to add or modify the Create Event widget. The Pipeline Canvas displays.
2. To add Create Event to an Evaluation, click **Add To**  , and then select **Create Event**  . Alternately, you can right-click on the Canvas to display the list of widgets, select **Create Event** and associate the Create Event widget with the Evaluation.
3. Click **Options** then click **Edit**  . The Create Event pane displays.
4. Under **Basic Configuration**, select the **Event Type** from the drop-down list.

   You can add additional Event Types, using the steps in Creating Event Types.

   You must define the Focus associated with events generated from this widget.

   - Select the required **Focus Table** from the drop-down list. The values which display are based on the tables you mapped in the Data Forge widget.
   - Select the required **Focus Column** from the drop-down list. The values which display are based on the columns you mapped in the Data Forge widget.

     > **Note**
     >
     > You can only select one value in each drop-down list. The column selected should be a column which is unique to the table.

   - Use the **Bind IDs Configuration** tab to choose which types of information you want to display in the generated events. You must select at least one Bind ID. For more information about this tab, see Configuring Bind IDs.
5. The **Highlights** tab displays any Highlights associated with this pipeline.

Highlights include the most salient facts associated with a match or alert, and are intended to aid with the understanding and possible disposition of an event. This tab includes the following details:

- Highlight Name
- New Highlight Name
- Column

You can add a new highlight by clicking **Add**, and following these steps:

a. Select the **Attribute** from the drop-down list.

b. Enter a **Name** for this highlight.

c. Click **Save**.

6. The **Bind IDs Configuration** tab displays the tables and columns which you have mapped in the Data Forge widget. Use the selector to choose the Bind IDs you are associating with the Focus of the events generated by this widget.

> **Note**
>
> You must select at least one Bind ID.

7. When you have finished configuring the Create Event widget, click **Save**.

Configuring Bind IDs

Bind IDs are used to define the focal entity by selecting the data on which the analysis is centered.

These bindings determine the types of information which display in the generated events. To configure Bind IDs, follow these steps:

1. Click to open the **Bind IDs Configuration** tab in the Create Event pane of the Create Event widget. The Bind IDs Configuration tab displays all tables and columns which were mapped in the data forge for this pipeline.
2. Use the Bind ID selector to choose which types of information you want to display in the generated events. For example, to display Customer information in your events, select the **Customer Internal ID** column from the Customer Dimension table as your Bind ID.

> **Note**
>
> Bind IDs selected should be a column which is unique to the table.

The following table provides examples of some commonly selected Bind IDs.

Table 2-12 Bind ID Examples

Focal Entity	Table Name	Column Name
Account	Account Dimension	Account Internal Identifier
Customer	Customer Dimension	Customer Internal Identifier
Transaction	Transactions	Transaction Internal ID
Account Profile	Account Summary	

3. Click **Save**.

3

About Scenario Pipelines

Scenario Pipelines enable you to create scenarios by defining behavior that consists of events in a predetermined order.

You can use these events to thread multiple data streams together. Scenarios are used to identify behaviors of interest, potentially problematic behaviors with respect to securities, regulations, and possible money-laundering activities.

These scenarios consider whether the geographical location or entities involved warrant enhanced scrutiny; monitor activity between accounts, customers, correspondents, and other entities to reveal relationships that could indicate efforts to launder funds; address sudden, significant changes in transaction activity that could indicate money laundering or fraud; and detect other types of activities that are considered potentially suspicious or indicative of money laundering.

As part of configuring a scenario pipeline, parameters are defined, which are then tuned in the Threshold Manager.

Pre-Configured DTP Scenario Pipelines

The application comes with the following ready-to-use, pre-configured scenario pipelines:

- CIB High Risk Geography Activity - Account Focus DTP
- Focal High Risk Entity - Customer Focus DTP
- High Risk Counter Party - Account Focus DTP
- Large Reportable Transaction - Customer Focus DTP

4

Managing Threshold Sets

FCCM Cloud Service uses tunable Thresholds to change variable values for scenarios.

When scenarios are created, thresholds are established. You can use the Threshold Editor to modify threshold values of scenarios, and create and edit threshold sets to fine-tune how that scenario finds matches, without changing the values defined at the data set or pattern level. These thresholds are applied to scenarios to find matches. Using this tool, you can enter a new value for a threshold (within a defined range) or reset the thresholds to their sample values.

Threshold sets allow you to run the same scenario multiple times against a variety of sources (for example, currencies, or jurisdictions) with separate threshold values for each source.

> **Note**
>
> Changing threshold values can generate significantly more or fewer alerts, depending upon the modifications made.

Accessing Threshold Sets

You can access the threshold sets through the Threshold Manager.

To access the All Threshold Sets page and view threshold sets, follow these steps:

1. In the Navigation List menu, select **Data Transformation Pipeline**.
2. Select **DTP Threshold Manager**. The All Threshold Sets page displays the complete list of threshold sets available in your implementation.
3. Select the check box for the threshold set you want to view, or click the threshold set name to view the threshold set details.

Creating a Threshold Set

You can create new threshold sets in the Threshold Editor.

To create a new threshold set, follow these steps:

1. Navigate to the **Threshold Editor** page.
2. Click **Add**  in the top right corner. The New Threshold Set page displays.
3. Enter the following values:
   a. Enter a **Name** for this threshold set.
   b. The scenario associated with this threshold set displays additional configurable parameters. These parameters are specific to the selected scenario.
   c. Select one or more **Jurisdictions** for this threshold set using the drop-down list. Jurisdiction refers to the division of data in the database based on criteria such as geographical boundaries, legal entities, and so on.

d. Select the **Period Type** used in this threshold set using the drop-down list. Options include Days or Months. By default, the period type is **Days**.

e. Enter the **Lookback Period** for this threshold set. Lookback period refers to the number of days or months to lookback from the current date or time to create a time window which is used to consider cases for correlation.

f. Enter the **Frequency Period** for this threshold set. Frequency period refers to how frequently the scenario should be run, by number of days or months.

g. Enter any **Comments** you have for this threshold set.

h. Enter the threshold values you want this threshold set to be updated with in the **New Value** field.

4.

- Click **Save** to save the values. A new threshold set is created and a message displays: *New Threshold Set created successfully*.
- Click **Save & Simulate**. A new threshold set is created, and the Create Simulator Conditions page displays. For more information about the Threshold Simulator, see Threshold Simulator.

Copying a Threshold Set

Copy an existing threshold set before modifying the values.

To copy and modify an existing threshold set, follow these steps:

1. Navigate to the Threshold Manager page.
2. Select the check box for the threshold set you want to copy.
3. Click **Copy** . The Copy Threshold Set window displays.
4. Enter the following values:
   a. Enter a **Name** for this threshold set.
   b. The scenario associated with this threshold set displays additional configurable parameters. These parameters are specific to the selected scenario.
   c. Select one or more **Jurisdictions** for this threshold set using the drop-down list. Jurisdiction refers to the division of data in the database based on criteria such as geographical boundaries, legal entities, and so on.
   d. Select the **Period Type** used in this threshold set using the drop-down list. Options include Days or Months. By default, the period type is **Days**.
   e. Enter the **Lookback Period** for this threshold set. Lookback period refers to the number of days or months to lookback from the current date or time to create a time window which is used to consider cases for correlation.
   f. Enter the **Frequency Period** for this threshold set. Frequency period refers to how frequently the scenario should be run, by number of days or months.
   g. Enter any **Comments** you have for this threshold set.
   h. Enter the threshold values you want this threshold set to be updated with in the **New Value** field.
5. Click **Save** to save the values.

Editing Threshold Sets

You can modify user-defined threshold sets in the Threshold Editor.

To edit a threshold set, follow these steps:

1. Navigate to the Threshold Editor page. The existing threshold sets are displayed.
2. Select the check box corresponding to the threshold set you want to edit.
3. Click **Edit**  . The Edit Threshold page is displayed.

   Alternatively, you can click **Edit** when viewing a threshold set in the View Threshold Set window.
4. Modify the required details.
   a. Select one or more **Jurisdictions** for this threshold set using the drop-down list. Jurisdiction refers to the division of data in the database based on criteria such as geographical boundaries, legal entities, and so on.
   b. Select the **Period Type** used in this threshold set using the drop-down list. Options include Days or Months. By default, the period type is **Days**.
   c. Enter the **Lookback Period** for this threshold set. Lookback period refers to the number of days or months to lookback from the current date or time to create a time window which is used to consider cases for correlation.
   d. Enter the **Frequency Period** for this threshold set. Frequency period refers to how frequently the scenario should be run, by number of days or months.
   e. Enter any **Comments** you have for this threshold set.
   f. Enter the threshold values you want this threshold set to be updated with in the **New Value** field.

   > **Note**
   >
   > You can modify the values of the existing parameters only, you cannot add new parameters.

5.
   - Click **Save**  to save the values. A new threshold set is created and a message displays: *New Threshold Set created successfully*.
   - Click **Save as New** to save the edited threshold set as a new threshold set. The New Threshold window opens. Enter a name for the threshold set and click **Save**. A new threshold set is created and a message displays: *New Threshold Set created successfully*.
   - Click **Save & Simulate**. A new threshold set is created, and the Create Simulator Conditions page displays. For more information about the Threshold Simulator, seeThreshold Simulator.

Deleting Threshold Sets

You can delete user-defined threshold sets in the Threshold Editor.

To delete a threshold set, follow these steps:

1. Navigate to the Threshold Editor page. The existing threshold sets are displayed.
2. Select the check box corresponding to the threshold set you want to delete.
3. Click **Delete**  . A message displays: *Are you sure you want to delete <threshold set>?*
4. Enter the reason for deletion in the **Your Comment** box. This is mandatory.
5. Click **Delete**. The threshold set is deleted.

About Threshold Simulator

The Threshold Simulator is used to run selected scenario pipelines against the selected threshold sets to find the matches obtained from these combinations.

These matches enable you to identify the events generated for the combination on a specified date. This can be helpful when you want to see which events would be generated with different threshold settings and be able to explain why your scenario is configured as it is, such as during audits.

View Simulator Conditions

You can view Simulator Conditions using the Threshold Manager.

To view simulator conditions for a threshold set, follow these steps:

1. In the Navigation List menu  , select **Data Transformation Pipeline**.
2. Select **DTP Threshold Manager**. The All Threshold Sets page displays.
3. In the All Thresholds page, select the check box corresponding to the threshold set you want to view simulation details for.
4. Click **Simulation History**  . The Simulation Details for this threshold set opens as a new tab.
5. Click the **Simulation Run ID** for the simulation you want to view conditions for. The View Threshold Set pop-up window displays the details of this threshold set.

   If you want to make modifications to these conditions, click **Edit & Simulate** . The Edit Threshold Set pop-up window displays. Follow the steps in Edit Simulator Conditions to make these changes.

Edit Simulator Conditions

You can edit Simulator Conditions using the Threshold Manager.

To edit existing simulator conditions for a threshold set, follow these steps:

1. In the All Thresholds page, select the check box corresponding to the threshold set you want to edit.

2. Click **Simulation History**  . The Simulation Details for this threshold set opens as a new tab.
3. Select the check box corresponding to the **Simulation Run ID** for the simulation you want to edit.
4. Click **Edit**  . The Edit Simulator Conditions pop-up window displays.

   Alternatively, you can access the Edit Simulator Conditions window by clicking **Edit & Simulate** in the View Simulator Conditions window.
5. To make modifications to these conditions, update the following values:
   a. Select one or more **Jurisdictions** for this threshold set using the drop-down list. Jurisdiction refers to the division of data in the database based on criteria such as geographical boundaries, legal entities, and so on.
   b. Enter the **Lookback Period** for this threshold set. Lookback period refers to the number of seconds, minutes, hours, or days to lookback from the current date or time to create a time window which is used to consider cases for correlation.
   c. Enter the **Frequency Period** for this threshold set. Frequency period refers to how frequently the scenario should be run.
   d. Enter the threshold values you want this threshold set to be updated with in the **New Value** field. Select the Batch Date as the date for the data you want to test the thresholds against. This can be the current date or a past date.
   e. Enter the Data Origin.
6. Click **Save & Simulate** to run the modified simulation. After running a simulation, the Simulator History window displays the result with the number of event matches and details of each event. You can view the following details:
   - Simulation ID: ID for this simulation run.
   - Data Origin: Data Origin of the data set the simulated scenario conditions are run against.
   - Jurisdiction: Jurisdiction the simulated scenario conditions are run for.
   - Batch Date: Date the simulated scenario conditions are run for.
   - Results: Number of event matches generated by the simulated scenario conditions during the run.
   - Event Details: Details for each event generated by the simulated scenario conditions during the run.
     – Event ID
     – Focus Name
     – Focus Type
     – Highlights

Creating New Simulator Conditions

You can create new Simulator Conditions using the Threshold Manager.

To create new simulator conditions for a threshold set, follow these steps:

1. In the **All Thresholds** page, select the check box corresponding to the threshold set you want to create new simulator conditions for.
2. Click **Simulation History**  . The Simulation Details for this threshold set opens as a new tab.
3. Click **Open Simulator**. The Create Simulator Condition pop-up window displays.

   Alternatively, you can access the Create Simulator window by clicking **Open Simulator** in the All Thresholds page. The Create Simulator Condition pop-up window displays.
4. Create the simulator conditions by providing the following details:
   a. Select the required scenario from the **Select Scenario** drop-down list.
   b. Select the required threshold set from the **Select Threshold** drop-down list.
   c. Select the **Batch Date** as the date for the data you want to test the thresholds against. This can be the current date or a past date.
   d. Enter the **Data Origin**.
5. Click **Save**.

Running Existing Threshold Sets

You can run a simulation for existing scenario threshold sets.

To run the threshold simulator for an existing scenario threshold set, follow these steps:

1. In the All Thresholds page, select the check box corresponding to the threshold set you want to run the simulator for.
2. Click **Open Simulator**. The Simulator Conditions pop-up window displays.

   Alternatively, you can also run existing threshold set from the Simulation Details tab by clicking Open Simulator. The Simulator Conditions pop-up window displays.
3. Provide the following details:
   a. Select the **Batch Date** as the date for the data you want to test the thresholds against. This can be the current date or a past date.
   b. Enter the **Data Origin**.
   c.
   d. Select one or more **Jurisdictions** for this threshold set using the drop-down list. Jurisdiction refers to the division of data in the database based on criteria such as geographical boundaries, legal entities, and so on.
   e. Enter the **Lookback Period** for this threshold set. Lookback period refers to the number of seconds, minutes, hours, or days to lookback from the current date or time to create a time window which is used to consider cases for correlation.
   f. Enter the **Frequency Period** for this threshold set. Frequency period refers to how frequently the scenario should be run.
   g. Enter any comments you have for this threshold set.
   h. Enter the threshold values you want for this threshold set in the **New Value** field.
4. Click **Save & Simulate** to run the modified simulation. After running a simulation, the Simulator History window displays the result with the number of event matches and details of each event. You can view the following details:

- Simulation ID: ID for this simulation run.
- Data Origin: Data Origin of the data set the simulated scenario conditions are run against.
- Jurisdiction: Jurisdiction the simulated scenario conditions are run for.
- Batch Date: Date the simulated scenario conditions are run for.
- Results: Number of event matches generated by the simulated scenario conditions during the run.
- Event Details: Details for each event generated by the simulated scenario conditions during the run.
  - Event ID
  - Focus Name
  - Focus Type
  - Highlights

Viewing Simulation Details

When a threshold set simulation has completed, you can view the details of all thresholds in this simulation and their results in the Simulation Details.

This allows you to determine whether your threshold set is generating the expected detection results or if further modification is required.You can also view the Simulation Details for all runs of a threshold set which has been previously run. To open the Simulation Details, follow these steps:

1. In the All Thresholds page, select the check box corresponding to the threshold set you want to view simulation details for.
2. Click **Simulation History**  . The Simulation Details for this threshold set opens as a new tab. You can view the following details:
   - Simulation Run: ID for this simulation run with the scenario name.
   - Jurisdiction: Jurisdiction the simulated scenario conditions are run for.
   - Run Date and Time: Date and time the simulation was run.
   - User ID: User who ran the simulation.

   You can export the results of the threshold set simulation runs in .xlsx format by selecting the check boxes for one or more simulation runs and clicking **Export Results**.
   You can delete a simulation run by selecting the check boxes for one or more simulation runs and clicking **Delete**. The Delete pop-up window displays: *Are you sure you want to delete <Simulation ID>*? Enter a comment and click **Delete**. A confirmation message displays.

   To return to the All Threshold Sets tab, click the **All Thresholds List** tab.

5

Using Jobs

The application uses jobs to define the instructions for executing the data pipelines or scenario pipelines against threshold sets, for example, running a scenario or loading data.

These jobs can be included in batches (groups of jobs) which run at configured intervals against the selected threshold to detect and generate events. This allows the jobs to run automatically, without requiring your involvement. Jobs can also be used to monitor the execution of jobs.

To access the DTP Jobs page, follow these steps:

1. In the Applications landing page, click the **Navigation Menu** to access the Navigation List. The Navigation List displays the list of modules.
2. Expand **Data Transformation Pipeline** in the Navigation List to view all DTP menu options.
3. Select **DTP Jobs**. The DTP Jobs page opens.

Creating Jobs

You can create new jobs to run in batches.

To create a new job, follow these steps:

1. Navigate to the Applications landing page.
2. Click the **Navigation Menu** to access the **Navigation List**. The Navigation List displays the list of modules.
3. Expand **Data Transformation Pipeline** in the Navigation List to view all DTP menu options.
4. Click **DTP Jobs** in the Navigation List. The Jobs page opens in a new window.
5. Click **Create Job** . The Create Job pane is displayed.
6. Provide the details as described in the following table:

Table 5-1 Fields to Create Jobs

Field	Description
Job Name	Enter the name for the job.
Pipeline Type	Select the pipeline type for which you want to create the job. The available options are Data and Scenario.
Pipeline	Select the pipeline from the drop-down list for which you want to create a job.

Table 5-1 (Cont.) Fields to Create Jobs

Field	Description
Threshold	Select the threshold set from the drop-down list. The drop-down list displays the list of thresholds that are created for the selected scenario pipeline. The job is run against the selected threshold to detect and generate events.

7. Click **Save** to save the changes. A new job is created and displayed in the Jobs page.

Editing Jobs

You can edit or delete user-defined jobs

To edit a job, follow these steps:

> **Note**
>
> Pre-configured jobs cannot be edited or deleted. You can only edit or delete user-defined jobs.

1. Navigate to the Applications landing page.
2. Click the **Navigation Menu** to access the **Navigation List**. The Navigation List displays the list of modules.
3. Expand **Data Transformation Pipeline** in the Navigation List to view all DTP menu options.
4. Click **DTP Jobs** in the Navigation List. The Jobs page opens in a new window.
5. Click on the Job that you want to modify and click **Options** then click **Edit**. The Edit Job pane displays.
6. Modify the required details in the Edit Job pane on the right-hand side.
7. Click **Save** to save the changes. The job is modified.

Deleting Jobs

You can edit or delete user-defined jobs.

To delete a job, follow these steps:

> **Note**
>
> Pre-configured jobs cannot be edited or deleted. You can only edit or delete user-defined jobs.

1. Navigate to the Applications landing page.

2. Click the **Navigation Menu** to access the **Navigation List**. The Navigation List displays the list of modules.
3. Expand **Data Transformation Pipeline** in the Navigation List to view all DTP menu options.
4. Click **DTP Jobs** in the Navigation List. The Jobs page opens in a new window.
5. Click on the Job that you want to modify and click **Options** ••• then click **Delete**. The job is deleted.

Viewing Execution Monitor

The Execution Monitor enables you to view the complete history of job execution, including the start and end time, status, log messages generated during job execution, and so on. This will help you see how your jobs are progressing and detect any recurring issues.

To view the execution monitor, follow these steps:

1. Navigate to the Applications landing page.
2. Click the **Navigation Menu** to access the **Navigation List**. The Navigation List displays the list of modules.
3. Expand **Data Transformation Pipeline** in the Navigation List to view all DTP menu options.
4. Click **DTP Jobs** in the Navigation List. The Jobs page opens in a new window.
5. On the Jobs page, click the job for which you want to view the execution monitor.
6. The Execution History pane displays the historical information of the selected job. The details include the Batch Run ID, the date and time during which a job is executed, status of the job execution, and so on.

   Use the **Task** and **Sort By** drop-down lists to manage how these details display.
7. Click **Monitor Execution** corresponding to the Batch Run ID for which you want to view more information.
8. The Execution Monitor page is displayed in a new window. The page contains details of the pipeline for which the job is created. The widgets in the pipeline are represented in different colors. Widgets are color coded to indicate the widget type and status of the job:
   - Completed
   - Failed
   - Pending
   - Not Started
9. Clicking **Log Messages**

   displays the Log Messages pane which lists a complete log of messages generated during the execution of the pipeline.
10. Click each widget to view the Log Messages dialog box with detailed information about the widget.

    The following information is included:

- **Log Messages**: Start and End Date
- **Query/REST Parameters**: The query that is being executed by this widget.
- **Parameters**: The relevant parameters being used in this widget.
- **Errors**: Details for any errors that occurred in this widget during Job execution.

6

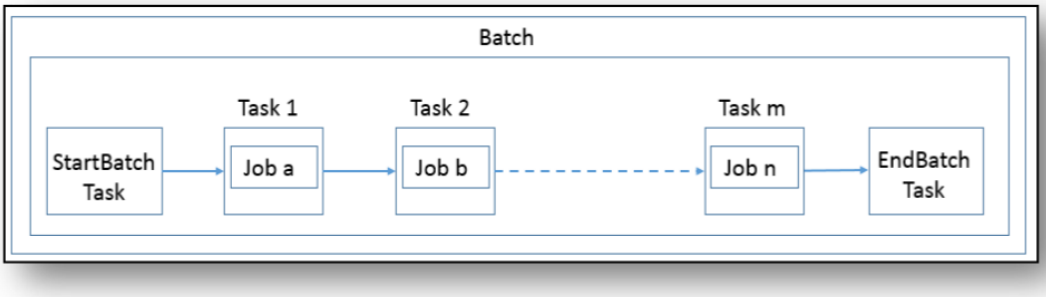
Managing Batches

A batch is a group of jobs that are scheduled to run at a defined interval of time, in sequence, automatically, without user involvement.

Oracle FCCM Cloud Service uses the Scheduler Service to create, schedule, execute and manage batches. A batch is a group of jobs that are scheduled to run at a defined interval of time, in sequence, automatically, without user involvement. Each batch begins with a StartBatch, includes any additional jobs that should be run in this batch, and then completes with the Endbatch.

> **Note**
>
> Oracle recommends that you first copy, customize, and then use the pre-configured batches as required.

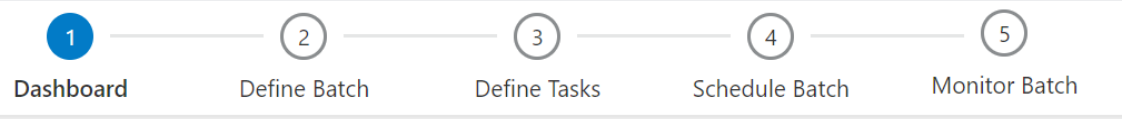
Figure 6-1 Flow of Batch



To execute the batches, use the Schedule Batch feature in the Scheduler Service. For more information, see Scheduler Service. You can use the Scheduler Service to first define the batch, then define which tasks should be included in this batch.

Next, you must schedule the batch. When the batch runs, you can monitor the batch to verify it is executing as intended. Click the Batch Scheduling Flow to navigate through the Scheduler Service.

Figure 6-2 Scheduler Service -- Batch Scheduling Flow



Pre-Configured Batches

The application contains certain pre-configured batches that can be used to run the default data. You must create new batches to run customer-specific data.

DTPTMScenario Batch Details

The following table provides the tasks that are configured for the DTPTMScenario batch. These tasks must be executed in the following order:

Table 6-1 DTPTMScenario Batch Details

Sequence	Tasks for DTPTMScenario Batch	Jobs for DTPTMScenario Batch	Pipelines for DTPTMScenario Batch
1	StartBatch	Not Applicable	Not Applicable
2	CIBHRGAC	CIB High Risk Geography Activity - Account Focus - DTP	CIB High Risk Geography Activity - Account Focus - DTP
3	FHRECU	Focal High Risk Entity - Customer Focus DTP	Focal High Risk Entity - Customer Focus DTP
4	LRTCU	Large Reportable Transaction - Customer Focus DTP	Large Reportable Transaction - Customer Focus DTP
5	EndBatch	Not Applicable	Not Applicable

7

Administrative Tasks

Certain tasks are frequently performed by Supervisor or Administrator users.

This section tells how to perform the following administrative tasks:

- Viewing Audit History
- Creating Focus Types
- Creating Event Types

Viewing Audit History

The Audit History provides a record of changes made to pipelines or threshold configuration.

For example, changing a pipeline name, adding or modifying widgets in the pipeline, creating or deleting a threshold set, changing threshold set configurations, and so on.

This allows you to view the changes made to a scenario before executing the scenario, and to detect and mitigate the risk of internal employee manipulation, as required by auditors and regulators.

You can also filter the results to show updates made to a specific pipeline since the last tuning cycle or last regulatory exam. If needed, you can export the Audit History data in .csv or .xls format to analyze further.

Accessing Audit History

You can access the Audit History from the Navigation List.

1. Navigate to the Applications landing page.
2. Click the **Navigation Menu**  to access the **Navigation List**. The Navigation List displays the list of modules.
3. Expand **Data Transformation Pipeline** in the Navigation List to view all DTP menu options.
4. Select **DTP Audit Screen**. The Audit History page displays.

   The following table describes the columns which display in the Audit History.

Table 7-1 Columns in the Audit History

Column	Description
Component	Type of component the action was taken on. For example, Data Transformation Pipeline or Threshold.
Component Name	Name of the threshold or pipeline the action was taken on. For example, if you are viewing the Audit History for a scenario pipeline, the scenario name will display.

Table 7-1 (Cont.) Columns in the Audit History

Column	Description
Sub Component	Type of sub component the action was taken on. For example, Business Filter, Aggregation, Evaluation, and so on. If there is no sub component, this column will appear blank.
Sub Component Name	Name of the sub component the action was taken on. For example, if a scenario Risk Indicator was updated, the name of the Risk Indicator will display, such as *Total of Very High Risk Amount Percentage*.If there is no sub component, this column will appear blank.
Action	Type of action that was taken. The DTP Audit History tracks the following types of actions: • Copy: An existing component has been copied with a new name. For example, an existing scenario pipeline is copied and then configured to meet your specific business needs. • Bulk Delete:A widget or pipeline is deleted at the parent level without deleting its underlying parameters. • Delete: A parameter or threshold is deleted. • Edit: A change has been made to an existing component or subcomponent. • Export:A pipeline or threshold set has been downloaded. • Import: A pipeline or threshold set has been imported. • Insert: A new threshold has been added to an existing pipeline.
Current State	The current state of the component which was acted upon. Compare the Current State with the Previous State to see the change which was made.
Previous State	The previous state of the component which was acted upon. Compare the Current State with the Previous State to see the change which was made.
Updated By	User who took the action.
Date & Time	Date and Time the action was taken.

Filtering Audit History

The Filter option allows you to search and narrow down the results of the Audit History.

You can use a combination of these search criteria to quickly find the components you are interested in. If you don't enter any value in any search field, it is equivalent to selecting all the criteria.
To filter the Audit History, follow these steps:

1. In the Audit History page, click **Filter** . The Filter criteria appear on the left-hand-side pane.
2. Select one or more criteria using the drop-down list selectors. You can filter by the following criteria:

- Updated by
- Action
- Component
- Component Name
- From Date
- To Date

3. Click **Apply** to filter the Audit History by the selected criteria.
   - Click **Reset** to clear the filter criteria.
   - Click **Close** to remove all filters.

Exporting Audit History

If needed, you can export the Audit History data in .csv or .xls format to analyze further.

Filter the Audit History data as desired, using the steps in Filtering Audit History.

To export the Audit History, follow these steps:

1. In the Audit History page, click **Export** to expand the output options.
2. Select from the following output options:
   - CSV
   - XLS
3. Select a folder to save the exported data.
4. Enter a file name, if desired.
5. Click **Save**. The exported Audit History is saved.

Creating Focus Types

Organizations can create and configure Focus Types according to their business needs.

A Focus Type describes the main entity on which the analysis, report, or calculation is centered. For example, a focal entity might be a customer whose transactions are being analyzed for suspicious activity. Focusing on the entity of interest ensures the granularity required for regulatory, risk, and business reporting or monitoring is appropriate. Relevant data is aggregated, filtered, or analyzed specifically for that entity.

The Account and Customer focus types are provided out-of-the-box, but you can create additional focus types to meet your organization's business needs.

To create a focus type, follow these steps:

1. Navigate to the Applications landing page.
2. Click the **Navigation Menu** to access the **Navigation List**. The Navigation List displays the list of modules.
3. Select **Data Transformation**, then select **Data Transformation Pipeline Admin**, then select **Focus Type**. The Focus Type admin page displays with a sortable list of current focus types.

4. Click **Add**  . The Add Focus Type pane opens.
5. Enter a **Name** for the Focus Type, such as External Entity. Valid characters include: A-Z a-z '-', with a maximum length of 100 Characters. For example, ABCDef or Abc-Def.
6. Click **Save**. A Focus Type ID is generated for this type and the new focus type is added to the list.

Creating Event Types

Organizations can create and configure Event Types according to their business needs.

Event Types, such as AML, show the designated classification for the behavior being detected when an event is generated.

To create an event type, follow these steps:

1. Navigate to the Applications landing page.
2. Click the **Navigation Menu** to access the **Navigation List**. The Navigation List displays the list of modules.
3. Select **Data Transformation**, then select **Data Transformation Pipeline Admin**, then select **Event Type**. The Event Type admin page displays with a sortable list of current event types.
4. Click **Add**  . The Add Event Type pane opens.
5. Enter a **Name** for the event type, such as Anti Money Laundering. Valid characters include: A-Z a-z '-', with a maximum length of 100 Characters. For example, ABCDef or Abc Def.
6. Enter a **Code** for the event type, such as AML.
7. Click **Save**. An Event Type ID is generated for this type and the new event type is added to the list.

   If you want to delete an Event Type, click **Delete**  . A verification message displays. Click **Delete** to remove the event type from the list.

Glossary

Index