CHAPTER 28

Workload Designer

Workload Designer

The **Workload Designer** portlet allows you to define, control, balance, and refine rules for managing workloads. Use the following features to manage workloads:

- Rulesets which are collections of related filters, throttles, events, states, and workload rules
- States which cause actions when a specific combination of planned environment and health condition occur
- Sessions limits, including query sessions limits, utility sessions limits, and utility limits
- Filters which reject queries
- Throttles which limit queries
- Exceptions which cause actions when one or more specific events occur
- Classification criteria which determine which queries use which rules

The **Workload Designer** portlet is a full-width portlet that cannot be minimized.

The **WORKLOAD DESIGNER** view provides controls to select a system and create, edit, view, and perform other actions on rulesets on that system.

The Workload Designer Ruleset views provide controls that allow you to refine settings for a system, states, sessions, filters, throttles, workloads, and exceptions.

About the Workload Designer View

The **WORKLOAD DESIGNER** view shows summary information about rulesets. Items in the options list depend on whether you are the ruleset owner. If a ruleset is locked by someone else, you have fewer options than if you are the ruleset owner. The **Working**, **Ready**, and **Active** sections contain the following options:

Working

Names and descriptions of rulesets that are being edited. In **Working**, you can create and import rulesets. Rulesets in **Ready** can be copied to **Working** for editing. Rulesets in **Working** can also appear in **Ready** and **Active**.

Ready

Rulesets that have been saved to the production system, but are not active. A ruleset must be in **Ready** before it is copied to **Active**. The **Active** ruleset cannot be deleted from **Ready**.

Active

Active ruleset on the production system. The only option available in the options list, if you have permissions, is to deactivate the ruleset.

Ruleset background color indicates synchronization between the **Working** and **Ready** sections. Gray in both sections indicates that the same version of the ruleset is in both sections. Blue indicates that the ruleset has been modified in the **Working** section since it was last saved to the **Ready** section.

TASM ruleset migration is performed automatically during the Teradata Database upgrade.

About Ruleset Options

The **WORKLOAD DESIGNER** view shows individual rulesets and specific options for each ruleset based on the permissions assigned to your role and the section in which the ruleset is located.

Working

The **Working** section contains rulesets that are being edited. Any ruleset listed in **Working** can also be listed in **Ready** and **Active**. Available options are:

- **Make Ready.** Copies the ruleset to the production server and to the **Ready** section.
- **Make Active.** Makes the ruleset the active ruleset on the production server. Copies the ruleset to the **Ready** and **Active** sections.
- **View.** Opens the ruleset for viewing.
- **Edit.** Opens the ruleset for editing.
- **Show All.** Lists all ruleset attributes on one page.
- **Lock.** Locks the ruleset so only the lock creator can edit the ruleset.
- **Unlock.** Unlocks the ruleset so others can edit the ruleset.
- **Clone.** Creates a copy of the ruleset. This option is useful if you want to use an existing ruleset as a base or template to create a ruleset.
- **Export.** Exports the ruleset XML file so you can view the file in a browser or save the file to a location you specify. Use with the **Import** button to copy a ruleset from one system to another.
- **Delete.** Removes the ruleset from the **Working** section.

Ready

The **Ready** section lists rulesets saved to the production server. Someone else can create a ruleset and add it to the **Ready** section. Then, from the **Ready** section, you can select **Copy to Working Rulesets** and edit the ruleset. Available options are:

- **Activate.** Makes the ruleset the active ruleset on the production server.
- **Copy to Working Rulesets.** Copies the ruleset to the **Working** section.
- **Delete.** Removes the ruleset from the production server.

Active

The **Active** section contains the active ruleset on the production server. If you have permission, the only available option is to **Deactivate** the current ruleset.

Creating a Ruleset

A ruleset is a complete collection of related filters, throttles, events, states, and workload rules. You can create multiple rulesets, but only one ruleset is active on the production server at a time. After creating a ruleset, you can specify settings, such as states, sessions, and workloads, using the toolbar buttons. New rulesets are automatically locked so only the owner can edit the ruleset.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.
2. Click **Create**.
3. Enter a ruleset name.
4. [Optional] Enter a description up to 80 characters long.
5. Click **Save**.
6. [Optional] Specify settings using the tabs in the **General** view, such as **Intervals** and **Bypass**.
7. [Optional] Click **States** and create a state matrix.
8. [Optional] Click **Sessions** and create any of the following:
   - **Query Sessions**
   - **Utility Limits**
   - **Utility Sessions**
9. [Optional] Click **Filters** and create a filter.
10. [Optional] Click **Throttles** and create a system throttle.
11. [Optional] Click **Workloads** and create a workload.
12. [Optional] Click **Exceptions** and create an exception.

Editing a Ruleset

In the **WORKLOAD DESIGNER** view, you can edit ruleset properties. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action. Without permission to edit the ruleset, the menu option is **View** and the ruleset view is read-only.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.
2. In the local **Working** section, click the ruleset name.
3. Specify settings using the toolbar buttons and tabbed views.
4. Click **Save** after making changes in each view.

Cloning a Ruleset

Cloning a ruleset makes an exact copy of the ruleset, except for the name and description.

Cloning is a convenient way to create a ruleset using the criteria of an existing ruleset as a base.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.

2. In the local **Working** section, click the ruleset and click **Clone**.

   An exact copy of the ruleset is made, and the **General** view appears.
3. Enter a name.
4. [Optional] Enter a description up to 80 characters long.
5. Click **Save**.
6. Specify additional settings using the toolbar buttons and tabbed views.
7. Click **Save**.

Deleting a Ruleset

Deleting a ruleset removes the ruleset and all associated information. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.
2. In the local **Working** section or the system **Ready** section, click the ruleset, and click **Delete**.
3. Click **Delete**.

   The ruleset is deleted from the section.

Importing a Ruleset

The import and export options can be used to copy a ruleset from one system to another. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action. Only rulesets exported from Workload Designer and a Teradata Database of the same release can be imported.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.
2. Click **Import**.
3. Enter a name for the imported ruleset.
4. Click **Browse**.
5. Locate and select the saved ruleset file.

   Note: Exported ruleset files might be stored in the download area configured for your browser.
6. Click **Save**.

Exporting a Ruleset

The import and export options can be used to copy a ruleset from one system to another. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action. Only rulesets exported from Workload Designer and a Teradata Database of the same release can be imported.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.

2. In the local **Working** section, click the ruleset , and click **Export**.
3. Click **Save**.

The ruleset file is saved to your download area or the location you specify, depending on your browser settings.

Showing All Criteria in a Ruleset

You can display a read-only summary of all settings and state-specific values for a single ruleset.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.
2. In the local **Working** section, click the ruleset , and click **Show All**.
3. [Optional] Do any of the following:
   - Click **Collapse All** to hide all sections of the view.
   - Click **Print** to print the full summary.
   - Click the section name to show or hide individual sections of the view.

Copying a Ruleset to the Production System

When you are finished editing a ruleset in the Working section, copy the ruleset to the **Ready** section on the production system. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action. After copying a ruleset to the **Ready** section, you can make the ruleset the active ruleset on the production system.

1. From the **WORKLOAD DESIGNER** view, select the production system containing the ruleset you want to copy.
2. In the **Working** section, click the ruleset , and click **Make Ready**.

Activating a Ruleset

Activating a ruleset copies the ruleset to the active state on the selected system. Only one ruleset is active on the system at a time. A ruleset activated from the local **Working** section is copied to the **Ready** section before being made active. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.
2. From a ruleset in the **Working** or **Ready** section, click the ruleset , and click **Make Active**.

Deactivating a Ruleset

Deactivating a ruleset removes the ruleset from the active state on the selected system. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.
2. In the system **Active** section, click the ruleset , and click **Deactivate**.

About Ruleset Locks

An exclusive lock can be placed on a ruleset so that the ruleset cannot be edited, deleted, or otherwise modified except by the owner of the lock. A ruleset is automatically locked by the

user when it is created and each time changes to the ruleset are saved. Use the **WORKLOAD DESIGNER** view to lock and unlock rulesets. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action.

The **WORKLOAD DESIGNER** view displays ruleset lock status:

Option	Description
	Locked.
	Unlocked.

The ruleset views display ruleset lock status:

Option	Description
	Unlocked.
	Locked by the current user.
	Locked by another user.

Locking or Unlocking a Ruleset

Locking a ruleset prevents others from editing or deleting the ruleset. The Teradata Viewpoint Administrator must grant your role permission to edit rulesets so you can complete this action.

1. From the **WORKLOAD DESIGNER** view, select a system from the list.
2. In the local **Working** section, click the ruleset , and click **Lock** or **Unlock**.

About the Ruleset General View

The ruleset **General** view displays general attributes of a ruleset and appears after you click the **General** button on the ruleset toolbar. The **General** view appears by default when creating, editing, cloning, or viewing a ruleset. This view contains the following tabs:

General

Ruleset name and description.

Intervals

Collection and reporting intervals.

Option	Description	Default Value	Minimum Value	Maximum Value
Event Interval	How often event thresholds are checked.	60 seconds	1 second	3600 seconds
Dashboard Interval	How often workload statistics are collected.	60 seconds	1 second	600 seconds

Option	Description	Default Value	Minimum Value	Maximum Value
Logging Interval	How often workload and exception logs are written.	600 seconds	1 second	3600 seconds
Exception Interval	How often exception thresholds are checked.	60 seconds	1 second	3600 seconds

Blocker

Settings for responding to throttled blockers. The log is located at DBC.TDWMEventLog. Selecting **Log** only logs the blocker. If **Abort** or **Release** is selected, the action is logged after the abort or release occurs.

Option	Description	Default Value	Values
Block Cycles	The number of intervals over which the query must be blocked before the specified **Block Action** is taken.	Off	Off 1 2 3
Block Action	The action to perform in response to query blocking.	Log	Log Abort (abort query) Release (release query from queue)

Activation

Features that are available when the ruleset is activated. **Events and States** is a required field and can be the only field selected.

Bypass

Users, accounts, and profiles whose queries are allowed to bypass system filters and throttles.

Option	Description
Source Type	Type of query source (you can select one or more items from each available source type): Usernames (usernames **dbc** and **tdwm** cannot be removed) Account Names Account Strings Profiles
Filter	Limits your choices in the source list.
Items	Lists the names of all available query sources of the selected type.
Bypass	Selected sources, listed by source type. Queries from each of the listed query sources will bypass system filters and throttles for this ruleset.

Defining System-Level Bypass Settings

A *system-level bypass* is a collection of users, accounts, and profiles that are not filtered or throttled at the system level. The **dbc** and **tdwm** usernames are listed automatically and cannot be changed, renamed, or removed.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **General**.
3. Click the **Bypass** tab.
4. Select a **Source Type** from the list to exclude it from system-level filters and throttles.
5. [Optional] Enter a filter string in the **Filter** box to limit your choices.
6. Select a source from the list or select multiple sources by using **Ctrl** or **Shift**.
7. Click **Add Selection** to add your selections to the bypass list.
8. [Optional] Repeat steps 4 through 7 to add sources to the bypass list.
9. [Optional] Mouse over a source in the bypass list, and click ⊠ to include it in system-level filters and throttles.
10. Click **Save**.

About Ruleset States

A *state* is the intersection of a health condition and a planned environment. A *health condition* is composed of unplanned events and a *planned environment* is composed of planned events. Creating states provides greater control over how the system allocates resources. When a health condition and a planned environment intersect, the resulting state triggers system changes.

Use the state matrix to create and organize states for a ruleset. With the state matrix, create states so one ruleset can respond to a range of different system conditions. For example, if you have many system users on weekdays, but run batch jobs on the weekend, allocate system resources differently during the week than you do on weekends by creating two planned environments: Weekdays and Weekends.

In the state matrix, create planned events, unplanned events, health conditions, planned environments, and corresponding states specific to your business situation. Update the state matrix at any time to reflect business, system, or priority changes.

The **Normal** health condition, **Always** planned environment, and **Base** state are defaults. The defaults apply unless planned or unplanned events occur, triggering other configured states. The defaults cannot be deleted or moved within the state matrix. Like any state, the **Base** state can be used in multiple cells of the matrix.

Any states you create use the default settings. The default settings can be viewed and edited on the state-specific settings tabs in workloads, filters, throttles, query sessions, and utility limits. You can override the default settings by entering new values on the state-specific settings tabs.

Using only a few states in the state matrix reduces maintenance time. However, consider adding states to the matrix to manage the following situations:

- Consistent, peak workload hours or days where priority management must be strictly assigned and enforced.
- Load or query times where priority tasks must finish within a specific time frame.
- Conditions where resources must be managed in a different way, such as giving higher priority to critical work when system health is degraded.

Creating a State Matrix

If you have created a ruleset, you can create a state matrix for the ruleset. The state matrix instructs the system as to which predefined state to use when a specific combination of planned environment and health condition exists.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **States**.

   The state matrix appears. For a new ruleset, the default state matrix consists of one cell.
3. Define at least one health condition.
4. Define at least one unplanned event.

   Unplanned events are system events, user-defined events, or a combination of these events.
5. Drag the unplanned event to the appropriate health condition so the event triggers the health condition.
6. Define at least one planned environment.
7. Define at least one planned event.

   Planned events are period events, user-defined events, or a combination of these events.
8. Drag the planned event to the appropriate planned environment so the event triggers the environment.
9. Define at least one state.
10. For each cell in the state matrix, drag and drop a state from the **States** list into the appropriate cell of the matrix. When the defined combination occurs, the state is triggered.

11 Click **Save**.

Defining Health Conditions

Health conditions define levels of system health and are used to reallocate system resources when an event degrades the system. When at least one unplanned event occurs, a health condition can be triggered. The default health condition is **Normal**, and it is used if no other health conditions are triggered. The **Normal** health condition always remains at the top. The lowest severity is listed at the top. The highest priority is at the bottom. If multiple unplanned events are active at the same time, the health condition with the highest severity is triggered.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **States**.

3 Mouse over **Health Conditions**, and click .

A health condition is added with the default name **newCond**.

4 Mouse over the health condition, and click .

5 Enter a name for the health condition.

6 Enter a minimum duration, in minutes, for the health condition.

The event that triggers the change to the health condition must remain above the trigger threshold for the minimum duration you enter before the health condition goes into effect. Setting minimum duration prevents short incidents of an event from triggering a change in the health condition.

7 Click **OK**.

8 [Optional] If there are more than two health conditions, click the health condition name, and drag the name to reorder.

9 [Optional] To delete a health condition, mouse over a health condition name, and click .

10 Click **Save**.

Defining System Events in the State Matrix

System events are unplanned events, such as a down node. Incorporating system events into health conditions within the state matrix gives you greater control over what actions Teradata Database takes when unexpected events occur. To create an event that only sends out a notification, create the event, but do not assign it to any unplanned environment. When the event occurs, the notification action you specified is triggered.

Note: Some event types give you the option to set qualification time. *Qualification time* is the length of time the specified event must last to be recognized as an event. Setting qualification time prevents very short incidents of an event from being acknowledged as events.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **States**.

3 Mouse over **Unplanned Events**, and click .

4 Click next to **Available Events**.

5 Select **System Event**.

6 Enter a name.

7 [Optional] Enter a description up to 80 characters long.

8 Select an **Event Type** from the list:

- **Node Down.** Specify the maximum allowed number of nodes down in a clique as a percentage between 0 and 100. The default is 24%.
- **AMP Fatal.** Specify the maximum number of AMPs reported as fatal at system startup.
- **Available AWT's.** Specify the minimum number of AWTs available on an AMP. Set a qualification time in minutes and seconds. (Release 13.10 and later.)
- **AWT Limit.** Specify the maximum number of AWTs in use on an AMP. Set the number of AMPs that must exceed the limit. Set a qualification time in minutes and seconds. (Release 12.0 and Release 13.0.)
- **Gateway Fatal.** Specify the maximum number of gateways reported as fatal at system startup.
- **PE Fatal.** Specify the maximum number of PEs reported as fatal at system startup.
- **Flow Control.** Specify the maximum allowed number of AMPs in flow control. Set a qualification time in minutes and seconds.

Maximum and minimum limits must be positive integers.

9 [Optional] Under **Notifications**, enable any of the following actions for the start or end of the event:

- **Send Alert.** Select the Teradata Alerts to run.
- **Run Program.** Select the program registered with Teradata Alerts to run.
- **Post to QTable.** Enter a string to post to the QTable at the start or end of the event. This option is not integrated with Teradata Alerts.

10 Click **OK**.

11 [Optional] Drag the event under a specific health condition.

12 Click **Close**.

13 Click **Save**.

Defining Planned Environments

Planned environments reallocate system resources during scheduled times. Planned environments are triggered when at least one planned event occurs. The default planned environment is **Always** and it cannot be deleted or moved. The order of precedence is from lowest to highest, reading from left to right. The planned environment with the highest precedence is activated if multiple planned events are active at the same time.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **States**.

3 Mouse over **Planned Environments**, and click  .

A planned environment is added with the default name **newEnv**.

4. Mouse over the planned environment, and click .
5. Enter a name.
6. Click outside the name.
7. If there are more than two planned environments, click a planned environment name, and drag the name to the left or right in the list to change the order of precedence.
8. [Optional] To delete a planned environment, mouse over the environment name, and click .

   You cannot delete the **Always** environment.
9. Click **Save**.

Defining Period Events in the State Matrix

Period events are planned events occurring on specific days and times, such as month-end financial processing. To create an event that only sends out a notification, create the event, but do not assign it to any planned environment. When the event occurs, the notification action you specified is triggered.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **States**.
3. Mouse over **Planned Events**, and click .
4. Click next to **Available Events**.
5. Select **Period Event**.
6. Enter a name.
7. [Optional] Enter a description up to 80 characters long.
8. [Optional] Select **Create New Corresponding Planned Environment** to automatically create a planned environment that this event triggers.
9. Select **Day of Week** or **Day of Month**, and click a single day or multiple days.
10. [Optional] Select **Month of Year**, and click one or more months.
11. [Optional] Select **Start Time**, and enter a start and end time.
12. [Optional] Select **Wrap around midnight** to have a time range spanning midnight for a period event.
13. [Optional] In **Notifications**, select any of the following:
    - **Send Alert.** Specify the Teradata Alerts to trigger at the start or end of the event.
    - **Run Program.** Specify the Teradata Alerts registered programs to trigger at the start or end of the event.
    - **Post to QTable.** Enter a string to post to the QTable at the start or end of the event. (This option is not integrated with Teradata Alerts.)
14. Click **OK**.
15. [Optional] Drag the event under a planned environment.

16 Click **Close**.

17 Click **Save**.

Wrap Around Midnight Example

When creating period events, use the **Wrap around midnight** option to have a time range spanning midnight.

If the **From** time of a period event is later than the **To** time, two time segments are available: midnight until the **To** time and the **From** time until midnight. When **Wrap around midnight** is not selected, the period event is in effect for segment 1 and segment 2 on each specified day. When **Wrap around midnight** is selected, the event is in effect for segment 2 on each specified day and for segment 1 on each day following the specified day.

For example, specify that a period event occurs on Mondays and Tuesdays with a **From** time of 17:00 and a **To** time of 08:00.

If **Wrap around midnight** is not selected:

	Monday	Tuesday	Wednesday
midnight—08:00 (time segment 1)	Yes	Yes	No
08:00—17:00	No	No	No
17:00—23:59 (time segment 2)	Yes	Yes	No

If **Wrap around midnight** is selected:

	Monday	Tuesday	Wednesday
midnight—08:00 (time segment 1)	No	Yes	Yes
08:00—17:00	No	No	No
17:00—23:59 (time segment 2)	Yes	Yes	No

Defining User-Defined Events in the State Matrix

User-defined events can be planned or unplanned. To create an event that only sends out a notification, create the event, but do not assign it to any planned or unplanned environment. When the event occurs, the notification action you specified is triggered.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **States**.

3 Mouse over **Unplanned Events** or **Planned Events**, and click .

4 Next to **Available Events**, click .

5 Select **User Defined Event**.

6 Enter a name.

7 [Optional] Enter a description up to 80 characters long.

8. [Optional] If you are creating the user-defined event as a planned event, you can select **Create New Corresponding Planned Environment** to automatically create a planned environment that this event triggers.
9. Under **Activate/Deactivate Event**, copy the appropriate SQL request text.
10. Paste the text into an SQL script.
11. [Optional] Under **Notifications**, do any of the following:
    - **Send Alert.** Specify the Teradata Alerts to trigger at the start or end of the event.
    - **Run Program.** Specify the Teradata Alerts registered programs to trigger at the start or end of the event.
    - **Post to QTable.** Enter a string to post to the QTable at the start or end of the event. (In Release 12.0, you can select the **Post to QTable** option, but the text box is not available.)
12. Click **OK**.
13. [Optional] Do one of the following:
    - For unplanned events, drag the event under a health condition.
    - For planned events, drag the event under a planned environment.
14. Click **Close**.
15. Click **Save**.

Defining Event Combinations in the State Matrix

An *event combination* is a mix of two or more different events, such as period, system, and user-defined events. Event combinations can be planned or unplanned. To create an event that only sends out a notification, create the event, but do not assign it to any planned environment. When the event occurs, the notification action you specified is triggered.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **States**.
3. Mouse over **Unplanned Events** or **Planned Events**, and click .
4. Next to **Available Events**, click .
5. Select **Event Combination**.
6. Enter a name.
7. [Optional] Enter a description up to 80 characters long.
8. [Optional] If you are creating the combination event as a planned event, you can select **Create New Corresponding Planned Environment** to automatically create a planned environment that this event triggers.
9. Add available events and operators to build the formula representing the combination of events that must occur to trigger the event. For example, Node Down OR Batch Processing, or NOT PEI.

   When creating event combinations, avoid placing two operators or two events next to each other. When an event combination is valid, the background of the **Event**

Combination Formula text box is white. If a combination is invalid, the background is orange.

10 [Optional] In **Notifications**, select any of the following:

- **Send Alert.** Specify the Teradata Alerts to trigger at the start or end of the event.
- **Run Program.** Specify the Teradata Alerts registered programs to trigger at the start or end of the event.
- **Post to QTable.** Enter a string to post to the QTable at the start or end of the event. (This option is not integrated with Teradata Alerts.)

11 Click **OK**.

12 [Optional] Do one of the following:

- For unplanned events, drag the event under a health condition.
- For planned events, drag the event under a planned environment.

13 Click **Close**.

14 Click **Save**.

Defining States in the State Matrix

You can create a state to control how resources are allocated in different health condition and planned environment combinations.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **States**.

3 Mouse over **States** at the top of the states list and click .

A state is added to the list with the default name **newState**.

4 Mouse over the state name, and click .

5 Enter a name.

6 Click outside the name.

7 [Optional] To edit a state, mouse over the state name, and click .

8 Click **Save**.

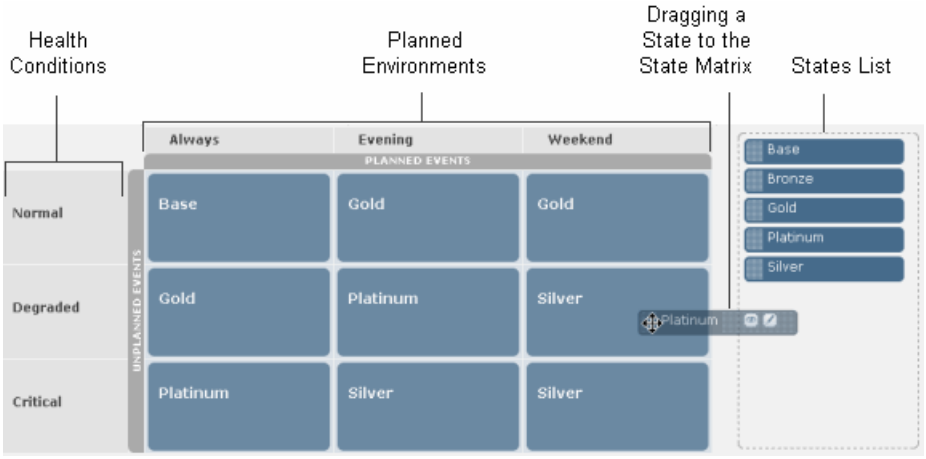
Mapping States in the State Matrix

After creating a state, you can map it in the state matrix to control how resources are allocated when different health condition and planned environment combinations occur. By default, the cell in the upper left corner of the state matrix is assigned the **Normal** health condition, the **Always** environment, and the **Base** state. This cell cannot be changed. All other cells in the matrix must be associated with a single state. Any state, including the **Base** state, can be used in multiple cells.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **States**.

3 To map a state in the state matrix, drag and drop a state name from the **States** list to a matrix cell.



4 [Optional] To view details about a state, mouse over the state name in the **States**, list and click .

5 Click **Save**.

Mapping Events in the State Matrix

If you have created events in the state matrix, you can combine the events with health conditions and planned environments for greater control of Teradata Database. Create the health conditions and planned environments that you need before mapping events in the state matrix.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **States**.

3 Map a planned event:

   a Mouse over **Planned Events**, and click .

   b In **Available Events**, drag and drop an event from the list into one or more of the **Planned Environment** columns.

   c [Optional] To create a planned event, click next to **Available Events**.

4 To map an unplanned event:

   a Mouse over **Unplanned Events**, and click .

   b In **Available Events**, drag an event from the list and drop it into one or more of the **Health Condition** rows.

   c [Optional] To create an unplanned event, click next to **Available Events**.

5 Click **Close**.

6 Click **Save**.

Deleting States from the State List

If a state is not used in the state matrix, you can remove it from the **State** list.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **States**.
3. In the **State** list, mouse over a state name, and click ⊠.
4. Click **Save**.

About Classification Settings

Workload Designer provides a common classification process for workloads, filters, throttles, query sessions, and utility sessions. Classification determines which queries use which rules. Teradata Database detects classification criteria before executing queries. The goal in creating a useful classification scheme is to meet business goals and fine-tune control of Teradata Database.

Modifications to the classification settings can be made in response to data monitoring, regular historical analysis, or changes. For example, classification groups may need to be created, or existing groups modified, if an application is added, two production systems are consolidated, or service-level goals are missed.

About Classification Criteria

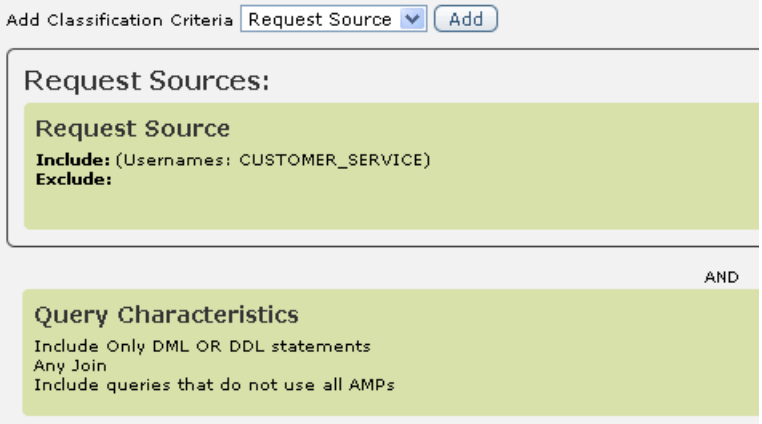
The following classifications are available:

- Request source (Where does the query come from?)
- Target (What is the query acting on?)
- Query characteristics (What is the query composed of?)
- Query band (What metadata is attached to the query?)
- Utility (Which utility submitted the query?)

A good approach to using classification is to first use request source to determine where the query is coming from. Often, the account string is selected, but other options include username, account name, or client IP address. If you need a more detailed level of classification, establish where the query data is located, such as a database, table, or view. To narrow classification further, select query characteristics, query bands, or utilities. For utilities, use the check boxes to select the specific utilities you want to include.

For example, you could create a filter and add the request source classification to reject all queries from the Finance department when the Red state is in effect. To further refine the filter, add the query characteristic classification to filter out all requests from the finance department that are estimated to run longer than 10 seconds. (You *include* classification items in a filter to reject those items.) The query characteristic setting is added to the request source setting already in place.

All classification settings for a workload, filter, throttle, query session, utility session, or utility limit are listed on the **Classification** tab.



On the **Classification** tab:

- Request source and target criteria can have a single criteria or several. In Release 13.10, if a criteria group has more than one criteria, then within the group the criteria are joined by AND. In Release 12.0 and Release 13.0, the criteria are joined by OR.
- If request source and target groups exist together, they are joined by an AND in Release 13.10. In Release 12.0 and Release 13.0, a button can be used to join the groups by AND or by OR.
- Within a query characteristics criteria group, if the **Statement Type** parameter is enabled, it is joined to any additional parameters with an AND. In Release 13.10, if there is more than one parameter, they are joined by an AND. In Release 12.0 and Release 13.0, a button can be used to join the parameters by AND or by OR.

About Request Source Classification Type

The request source classification type establishes which username, account name, account string, profile, application, client IP address, or client ID is making the request.

Consider the following when using request source to classify information:

- A source type can only be used once per rule. After a source type is used, it no longer appears in the menu.
- A match string must be an exact match. A match string can contain ? to match exactly one character or an * to match zero or more characters. Use the **Match String Include** and **Exclude** buttons to add the match string to a list. You can also select items from the list and use the **Items Include** and **Exclude** buttons to create your classification criteria.

Adding Request Source Classification Type

You can classify filters, throttles, workloads, utility sessions, and query sessions by request sources such as account name or client IP address.

In Release 12.0 and Release 13.0, when adding the request source classification type to filters and throttles, an **Add** button is available instead of the **Include** and **Exclude** buttons. In Release 12.0, when adding the request source classification type to workloads, all specified criteria can be either included or excluded. In Release 12.0, wild cards are not available in **Match String**. In Release 12.0 and Release 13.0, the request source types available for filters and throttles are **Username**, **Profile**, **Account Name**, and **Account String**.

1. Click **Filters**, **Throttles**, **Workloads** or **Sessions** (**Query Sessions** or **Utility Sessions** tab) and select the name of a rule or create a rule.
2. Click the **Classification** tab.
3. Do one of the following:
   - From the **Add Classification Criteria** list, click **Request Source**, and click **Add**.
   - Click **Add Criteria** if you are classifying a query session.
   - Select an existing request source criteria.
4. Select a **Source Type** from the list.
5. Do at least one of the following:
   - In **Match String**, enter a string. A match string can contain ? to match exactly one character or an * to match zero or more characters. (Wildcards in match strings are supported in Release 13.0 and later.) Use the **Match String Include** and **Exclude** buttons to add the match string to a list.
   - Select **Items** from the list and use the **Items Include** and **Exclude** buttons to create your classification criteria. (In Release 12.0 and Release 13.0, use the **Add** button.)
6. Click **OK**.

About Target Classification Type

The target classification type establishes the query data location.

Consider the following when using target to classify information:

- Available target types include database, table, macro, view, or stored procedure. If table, macro, view, or stored procedure is selected, a database selection list appears. A target type can only be used once per rule. After a target type is used, it no longer appears in the menu.
- Optionally, each selected target item can have subcriteria. For example, if you select a database as the target, you could add subcriteria so that it only applies if you are performing a full table scan. If you select two or more subcriteria, they must all be present for the classification setting to be used. The icon  appears next to target items containing subcriteria.
- A match string must be an exact match. A match string can contain ? to match exactly one character or an * to match zero or more characters. Use the **Match String Include** and **Exclude** buttons to add the match string to a list. You can also select items from the list and use the **Items Include** and **Exclude** buttons to create your classification criteria.

Adding Target Classification Type

The target classification type specifies the query data location. You can classify filters, throttles, or workloads by targets such as database, table, or stored procedure. You can add subcriteria in Release 13.10. If you add multiple subcriteria to a single item, all subcriteria conditions must be true in order for the query to be classified into the rule. In Release 12.0 and Release 13.0, when adding the target classification type to filters and throttles, an **Add** button is available instead of the **Include** and **Exclude** buttons. In Release 12.0, when adding the target classification type to workloads, all specified criteria can be either included or excluded. In Release 12.0, wild cards are not available in **Match String**.

1. Click **Filters**, **Throttles**, or **Workloads** and select the name of an existing item or create one.
2. Click the **Classification** tab.
3. Do one of the following:
   - From the **Add Classification Criteria** list, click **Target**, and click **Add**.
   - Select an existing target criteria.
4. Select a **Target Type** from the list.
5. [Optional] Select a database from the list for **Table**, **View**, **Macro**, or **Stored Procedure** target types.
6. Do at least one of the following:
   - Enter a **Match String** and use the **Match String Include** and **Exclude** buttons to add the match string. A match string can contain ? to match exactly one character or an * to match zero or more characters.
   - Select **Items** from the list and use the **Items Include** and **Exclude** buttons to add the items.
7. [Optional] To add subcriteria, mouse over an item in the **Selected** list, click  , and choose from the following:
   - **Full Table Scan.** Include or exclude full table (all row) scans.
   - **Join Type.** Select a type, such as **No Join** or **Any Join**.
   - **Minimum Step Row Count.** Set minimum rows at each step.
   - **Maximum Step Row Count.** Set maximum rows at each step.
   - **Minimum Step Time.** Set minimum time at each step.
8. If you specified subcriteria, click **Apply**.

   The icon  appears next to target items containing subcriteria.
9. Click **OK**.

About Query Characteristic Classification Type

The *query characteristic* classification type describes a query by answering such questions as what does the query do and how long will the query run.

Consider the following when using query characteristics to classify information:

- After a characteristic is selected, its value can be edited.
- Many characteristics have minimum and maximum values that can be set independently. You can set all values above the minimum, below the maximum, or between a minimum and a maximum.
- Query characteristic classification and utility classification are mutually exclusive. If you use one, the other option is not available.
- You can have one query characteristic classification per rule.
- If you select **Join Type**, you can choose from **No Join**, **Any Join**, **Product Join**, **No Product Join**, **Unconstrained Product Join**, and **No Unconstrained Product Join**.

Adding Query Characteristic Classification Type

You can classify filters, throttles, or workloads by query characteristics.

1. Click **Filters**, **Throttles**, or **Workloads** and select the name of an existing item or create one.
2. Click the **Classification** tab.
3. Do one of the following:
   - From the **Add Classification Criteria** list, click **Query Characteristics**, and click **Add**.
   - Select an existing query characteristics criteria.
4. Choose from the following query characteristics criteria:
   - **Statement Type.** Click **DDL**, **DML**, or **SELECT**.
   - **AMP Limits.** Include or exclude queries that use all AMPs. (Available for filters in Release 13.10.)
   - **Step Row Count.** Set minimum and maximum rows at each step.
   - **Final Row Count.** Set minimum and maximum rows in the result set.
   - **Estimated Processing Time.** Set minimum and maximum estimated processing time. (Longer or more complex queries have less accurate estimates.)
   - **Minimum Step Time.** Set a minimum time at each step.
   - **Join Type.** Use the **Include Only** list to select one of the following join types: **No Join**, **Any Join**, **Product Join**, **No Product Join**, **Unconstrained Product Join**, or **No Unconstrained Product Join**.
   - **Full Table Scan.** Choose to include or exclude full table (all row) scans.
5. Click **OK**.

About Query Band Classification Type

The query band classification type describes the query band data attached to a query.

Consider the following when using query band to classify information:

- A name must be selected from the **Name** list or entered into the box.
- After picking a name, one or more values must be specified. The value is selected from the **Previously Used Values** list or entered into the **New Value** box. Multiple values can be selected for the same name.
- After a name and value are specified, the **Include** and **Exclude** buttons are available.
- Multiple included query band key and value pairs are connected with "AND."
- Multiple excluded query band key and value pairs are connected with "OR."

Adding Query Band Classification Type

A query band contains name and value pairs that use predefined names (on Teradata Database) or custom names to specify metadata, such as user location or application version. The query band classification type describes the query band data attached to a query.

In Release 12.0 and Release 13.0, when adding the query band classification type to filters and throttles, an **Add** button is available instead of the **Include** and **Exclude** buttons. In

Release 12.0, when adding the query band classification type to workloads, all specified criteria can be either included or excluded.

1. Click **Filters**, **Throttles**, or **Workloads** and select the name of an existing item or create one.
2. Click the **Classification** tab.
3. Do one of the following:
   - From the **Add Classification Criteria** list, click **Query Band**, and click **Add**.
   - Select an existing query band criterion.
4. Do one of the following:
   - Select a predefined query band name from the list.
   - Enter a name.
5. Select a **Previously Used Value** or enter a **New Value**.

   You must select a name and a value.
6. Use the **Include** and **Exclude** buttons to create your classification criteria.
7. Click **OK**.

About Utility Classification Type

The utility classification type identifies which utility submitted the query.

Consider the following when using utility to classify information:

- Available utility types include **FastLoad**, **FastExport**, **MultiLoad**, and **Archive/Restore**. Select a top level utility such as **FastExport** or a specific implementation of a utility such as **JDBC FastExport**.
- Utility classification and query characteristic classification are mutually exclusive. If you use one, the other option is not available.
- You can have one utility classification per rule.

Adding Utility Classification Type

You can classify filters, throttles, workloads, or sessions by utilities such as FastLoad or MultiLoad.

1. From the **Filters**, **Throttles**, **Workloads**, or **Sessions** (click the **Utility Limits** or **Utility Sessions** tab) button, select the name of an item or create one.
2. Click the **Classification** tab.
3. Do one of the following:
   - From the **Add Classification Criteria** list, click **Utility**, and click **Add**.
   - Select an existing utility criteria.
4. Select any combination of **FastLoad**, **FastExport**, **MultiLoad**, and **Archive/Restore** utilities. Select a utility type, such as **FastLoad** or **FastExport**, or a specific version, such as **JDBC FastLoad** or **Stand Alone MultiLoad**.
5. Click **OK**.

Deleting a Classification Type

You can delete classification criteria from filters, throttles, workloads, query sessions, utility sessions, or utility limits at any time.

1. From the ruleset toolbar, click **Filters**, **Throttles**, **Workloads**, or **Sessions** (click the **Query Sessions**, **Utility Limits**, or **Utility Sessions** tab).
2. In **Name**, select existing item.
3. Click the **Classification** tab.
4. Mouse over an existing classification criteria, and click ⊠. If you are deleting a utility limit criterion, clear the utilities you want to delete from the classification.
5. Click **Save**.

About Ruleset Sessions

This topic describes the session limit information you can specify when creating and editing rulesets. In Release 12.0 and Release 13.0, session throttles are configured on the **Throttles** tab. The **Sessions** view appears after you click the **Sessions** button on the ruleset toolbar and has the following tabs:

Query Sessions

Limits on the number of query sessions that can be logged on at one time. You can create, enable, clone, and delete query sessions. (Release 13.10 and later)

Query Sessions by State

Limits on the number of query sessions for each state. The default session limit for a state is listed, along with each state you have created and its assigned, state-specific session limit. (Release 13.10 and later)

Utility Limits

Limits on the number of utilities that can run at the same time. You can create, enable, clone, and delete utility limits.

Utility Limits by State

Limits on the number of utilities for each utility limit rule in each state. The default utility limit for a state is listed, along with each state you have created and its assigned, state-specific utility limit.

Utility Sessions

Limits on the number of sessions a specific utility can use. You can create, enable, clone, and delete utility sessions. (Release 13.10 and later)

Utility Sessions Evaluation Order

Precedence, from highest to lowest, of utility session rules. Evaluation order determines the rule in which the utility job is placed if a utility job matches more than one utility session rule. (Release 13.10 and later)

Creating a Query Session

You can create a query session in Release 13.10 or later. In Release 12.0 and Release 13.0, use a throttle. A *query session* limits the number of sessions that can be logged on at one time. After the query session is created, additional controls in the **Query Sessions** tab allow you to clone, delete, and enable or disable the query session. View all created query sessions on the **Query Sessions by State** tab.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Sessions**.
3. Click the **Query Sessions** tab.
4. Click **Create Query Session**.
5. Enter a name.
6. [Optional] Enter a description up to 80 characters long.
7. Select a **Rule Type** from the list:
   - Select **Collective** if you want all users that meet the classification criteria treated as a group, with the group allowed a maximum number of queries.
   - Select **Individual** if you want to apply limits to each user individually.
   - Select **Member** if you want accounts or profiles that represent user groups used as the classification criteria for the rule. Limits are placed on each individual in the group, and no limit is placed on the account or group.
8. Click **Save**.
9. Click the **Classification** tab.
10. Add classification criteria.
11. Click the **State Specific Settings** tab.
12. Set state-specific session limits.

Setting Classification for Query Sessions

Add classification settings to existing query sessions or when creating a query session. Classification options determine which sessions match a rule.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Sessions**.
3. Do one of the following in the **Query Sessions** tab:
   - In **Name**, select an existing query session.
   - Click **Create Query Session**, enter a name and optional description (up to 80 characters long), select a rule type, and click **Save**.
4. Click the **Classification** tab.
5. Click **Add Criteria**.
6. In **Source Type**, click and select one of the following:

- **Account Name.** The Teradata Database account name.
- **Account String.** The Teradata Database account identification string.
- **Profile.** The Teradata Database profile name.
- **Application.** The application on the network client.
- **Client IP Address.** The IP address of the network client.
- **Client ID.** The logon name on the network client.

7 Do at least one of the following:

- In **Match String**, enter a string. A match string can contain ? to match exactly one character or an * to match zero or more characters. Use the **Match String Include** and **Exclude** buttons to add the match string to a list.
- Select **Items** from the list and use the **Items Include** and **Exclude** buttons to create your classification criteria.

8 Click **OK**.

Setting State Specific Values for Query Sessions

Add state-specific settings to existing query sessions or when creating a query session.

You can override the default by specifying session limits on a per-state basis. For example, you might want to have session limits during high-traffic states and no session limits during low-traffic states. View all created query sessions on the **Query Sessions By State** tab.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Sessions**.

The **Query Sessions** tab appears, listing all existing query sessions and their attributes.

3 Do one of the following in the **Query Sessions** tab:

- In **Name**, select an existing query session.
- Select **Create Query Session**, enter a name and optional description (up to 80 characters long), select a rule type, and click **Save**.

4 Select **State Specific Settings**.

5 Click a state in the state matrix to set a query session limit for that state.

6 Select **Create State Specific Settings**.

7 Select **Unlimited**, or enter a session limit in the box.

8 Click **OK**.

Your selection is applied to each cell having that state, and overrides the setting specified in **Default Settings**.

9 [Optional] Change the default setting by selecting **Unlimited**, or enter a session limit number in the box.

10 Click **Save**.

Creating a Utility Limit

A *utility limit* determines the number and type of utility jobs that can be run at one time. After the utility limit is created, additional controls in the **Utility Limits** tab allow you to clone, delete, and enable or disable the utility limit.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Sessions**.
3. Click the **Utility Limits** tab.
4. Click **Create Utility Limit**.
5. Enter a name.
6. [Optional] Enter a description up to 80 characters long.
7. Select **Reject utilities that exceed limit** to abort any utilities that exceed the limit. If you do not select this option, utilities are delayed.

   (Release 12.0 only.)
8. Click **Save**.
9. Click the **Classification** tab.
10. Select the utilities to which the limit should be applied.

    When a utility limit is created, several utilities are selected by default. The default utilities can be cleared.
11. Click **Save**.
12. Click the **State Specific Settings** tab.
13. Click **Save**.

Setting Classification for Utility Limits

Add classification settings to existing utility limits or when creating a utility limit. Classification options determine which sessions match a rule.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Sessions**.
3. Click the **Utility Limits** tab.
4. Do one of the following:
   - In **Name**, select an existing utility limit.
   - Click **Create Utility Limit**, enter a name and optional description (up to 80 characters long), and click **Save**.
5. Click the **Classification** tab.
6. Select any combination of **FastLoad**, **FastExport**, **MultiLoad**, and **Archive/Restore** utilities. Select a utility type such as **FastLoad** or **FastExport**, or a specific version, such as **.NET FastLoad** or **Stand-Alone MultiLoad**. (In Release 12.0 and Release 13.0, specific versions are not available, and only one utility type can be selected.)

When a utility limit is created, several utilities are selected by default. The default utilities can be cleared.

7 Click **Save**.

Setting State-Specific Job Limits for Utility Limits

Set state-specific job limits for existing utility limits or when creating a utility limit.

You can override the default by setting job limits on a per-state basis. For example, you might want to raise the job limit during a low-traffic state, and lower the job limit during a high-traffic state.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Sessions**.

3 Select **Utility Limits**, and do one of the following:

- In **Name**, select a utility limit.
- Select **Create Utility Limit**, enter a name and optional description (up to 80 characters long), and click **Save**.

4 Select the **State Specific Settings** tab.

5 Click a state in the state matrix to select it.

6 Select **Create State Specific Settings**.

7 Select **Job Limit**.

8 Enter a number in the box.

9 Select **Delay** or **Reject**.
(Release 13.0 or later.)

10 Click **OK**.

Your selection is applied to each cell having that state, and overrides the settings specified in **Default Values**.

11 [Optional] To change the default setting:

a Select **Job Limit**.

Your selection is applied to each cell having that state, and overrides the settings specified in **Default Values**.

b Enter a number in the box.

c Select **Delay** or **Reject**. (Release 13.0 or later.)

12 Click **Save**.

Creating a Utility Session

You can create a utility session in Release 13.10 or later. A *utility session* controls the number of sessions that are allowed to be logged on to each utility at one time. After the utility session is created, additional controls in the **Utility Sessions** tab allow you to clone, delete, and enable or disable the utility session.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Sessions**.
3. Click the **Utility Sessions** tab.
4. Click **Create Utility Session**.
5. Enter a name.
6. [Optional] Enter a description up to 80 characters long.
7. Select the utilities to which this session limit applies.
8. From the list, select the **Data Size**.
9. In **Max Sessions**, enter the maximum number of sessions that are allowed to be logged on to each of the selected utilities at one time.
10. Click **Save**.
11. Click the **Classification** tab to add and save classification criteria.

Setting Classification for Utility Sessions

Add classification settings to existing utility sessions or when creating a utility session. Classification options determine which sessions match a rule.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Sessions**.
3. Click the **Utility Sessions** tab.
4. Do one of the following:
   - In **Name**, select an existing utility session.
   - Click **Create Utility Session**, enter a name, select the utilities to which the session limit applies, select a **Data Size**, enter a **Max Sessions** number, and click **Save**.
5. Click the **Classification** tab.
6. In the **Add Classification Criteria** list, select **Request Source** or **Query Band**.
7. Click **Add**.
8. Specify options based on the classification criteria you selected. If you selected **Request Source**, do at least one of the following:
   - In **Match String**, enter a string. A match string can contain **?** to match exactly one character or an * to match zero or more characters. Use the **Match String Include** and **Exclude** buttons to add the match string to a list.
   - Select **Items** from the list and use the **Items Include** and **Exclude** buttons to create your classification criteria.
9. Click **OK**.

Setting Evaluation Order for Utility Sessions

You can create utility sessions and set evaluation order in Release 13.10 or later. If a utility job matches more than one utility session rule, evaluation order determines the rule in which the utility job is placed. The rule in the highest position on the **Utility Sessions Evaluation**

Order tab is applied. You cannot change the order of the system rules located at the bottom of the list.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Sessions**.
3. Click the **Utility Sessions Evaluation Order** tab.
4. Drag rules to reorder the list of utility rules.
5. Click **Save**.

About Ruleset Filters

A *filter* rejects or limits a query before the query starts running. You can specify filters when creating or editing a ruleset. Add criteria to each filter that identifies the queries to which the filter should be applied. When creating or editing a ruleset, filters are specified by clicking **Filters** in the ruleset toolbar. Any time after creating a filter, you can specify the states to which you want the filter applied.

Following are examples of using filters:

- Create a filter that prohibits a specific user from running a query with an estimated processing time of longer than 15 minutes.
- Create a filter to limit all members of a specific department that runs large reports from accessing the database during peak work hours.

Creating Filters

A filter rejects or limits user queries.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Filters**.
3. Click **Create Filter**.
4. Enter a name.
5. [Optional] Enter a description (up to 80 characters long).
6. [Optional] Select **Global Rule - applies to all requests** to apply global filters to all objects and, as a result, to all logon and query requests that meet the criteria of the rule. If **Global Rule - applies to all requests** is selected, the request source and target classification types are not available. If request source or target classification type is used, the **Global Rule - applies to all requests** option is not available.

   (Release 12.0 and Release 13.0 only.)
7. [Optional] Select **Warning Only** to have a warning message for the filter logged by the database. (Queries still run.)
8. Click **Save**.
9. Click the **Classification** tab.
10. Add and save classification criteria.

Include classification items in a filter to reject those items. For example, create a filter and add classification criteria to reject all queries from the finance department.

11 Click the **State Specific Settings** tab.

12 Enable or disable the filter for specific states.

13 Click **OK**.

14 Click **Save**.

Filter Classification Criteria

Some filters for query characteristics classification criteria are only available in Release 13.10.

Filter Options	Rel 12.0	Rel 13.0	Rel 13.10
Statement Type	available	available	available
AMP Limits			available
Step Row Count	available	available	available
Final Row Count	available	available	available
Estimated Processing Time Rel	available	available	available
Minimum Step Time			available
Join Type	available	available	available
Full Table Scan	available	available	available
AND or OR option to establish how multiple criteria are joined	available	available	

Setting Classification for Filters

Filters limit or reject user queries. Add classification settings to existing filters or when creating a filter. Classification options determine which queries or sessions match a rule.

When using the query characteristics classification criteria with filters, consider the following:

- In Release 13.0 and earlier, select the **AND** or **OR** option to establish how multiple criteria are joined.
- In Release 13.0 and earlier, the **Statement Type**, **AMP Limits**, and **Estimated Processing Time** criteria are available.
- In Release 13.10 or later, a **Minimum** option is available when using **Step Row Count** criteria.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Filters**.

3 Do one of the following:

   - In **Name**, select an existing filter.

- Click **Create Filter**, enter a name and optional description (up to 80 characters long), and click **Save**.

4. Click the **Classification** tab.
5. In the **Add Classification Criteria** list select **Request Source**, **Target**, **Query Characteristics**, **Query Band**, or **Utility**.
6. Click **Add**.
7. Specify options based on the classification criteria you selected.
8. If you selected **Request Source** or **Target**, do at least one of the following:
   - In **Match String**, enter a string. A match string can contain **?** to match exactly one character or an * to match zero or more characters. Use the **Match String Include** or **Exclude** buttons to add the match string to a list.
   - Select **Items** from the list and use the **Items Include** and **Exclude** buttons to create your classification criteria.
9. Click **OK**.

Setting State-Specific Values for Filters

Add state-specific settings to existing filters or when creating a filter.

You can override the default by enabling or disabling the filter on a per-state basis. For example, you may want to leave a filter enabled under all circumstances except when a specific state occurs.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Filters**.

   The **Filters** tab appears, listing all existing filters and their attributes.
3. Do one of the following in the **Filters** tab:
   - In **Name**, select an existing filter.
   - Select **Create Filter**, enter a name and optional description (up to 80 characters long), and click **Save**.
4. Select **State Specific Settings**.
5. Click a state in the state matrix to enable or disable the filter for that state.
6. Select **Create State Specific Setting**.
7. Select **Filter is Enabled** or **Filter is Disabled**.
8. Click **OK**.

   Your selection is applied to each cell having that state, and overrides the settings specified in **Default Settings**.
9. [Optional] Change the default setting by selecting **Filter is Enabled** or **Filter is Disabled**.
10. Click **Save**.

About Ruleset Throttles

A *throttle* limits the number of user queries that can be active at one time. When creating or editing a ruleset, you can specify throttles. Throttles are different from filters, which reject queries. After the throttle limit is reached, the workload adds new queries to the delay queue.

The following are examples of using throttles:

- Create a throttle that limits a specific user to running no more than 10 queries at a time.
- Create a throttle that limits a specific department to no more than 4 simultaneous queries.

When creating or editing a ruleset, throttles are specified by clicking **Throttles** in the ruleset toolbar.

Anytime after creating a ruleset throttle, you can specify the throttle limits for each state.

Creating Throttles

Throttles limit the number of user queries that can be active at one time. This is different from filters, which reject queries and prevent them from running at all.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Throttles**.
3. Click **Create Throttle**.
4. Enter a name.
5. [Optional] Enter a description up to 80 characters long.
6. Select a **Rule Type**:
   - Select **Collective** if you want all users that meets the classification criteria treated as a group, with the group allowed a maximum number of queries.
   - Select **Individual** if you want to apply limits to each user individually.
   - Select **Member** if you want accounts or profiles that represent user groups used as the classification criteria for the rule. Limits are placed on individuals in the group and no limit is placed on the account or group.
   - Select **Global** if you want all queries placed in a single queue. If **Global** is selected, the request source and target classification types are not available. If request source or target classification type is used, the **Global Rule** option is not available.(Release 12.0 and Release 13.0.)
7. [Optional] Select **Disable Manual Release or Abort** to prevent Teradata Database Administrators from aborting or releasing throttled queries in the queue.
8. [Optional] Select **Reject throttled queries** to abort throttled queries. (Release 12.0.)
9. Click **Save**.
10. Click the **Classification** tab.
11. Add and save classification criteria.
12. Click the **State Specific Settings** tab.
13. Define query limits for specific states.

14 Click **OK**.

15 Click **Save**.

Throttle Classification Criteria

Some throttles for query characteristics classification criteria are only available in Release 13.10.

Filter Options	Rel 12.0	Rel 13.0	Rel 13.10
Statement Type	available	available	available
AMP Limits	available	available	available
Any Step Time Exceeds	available	available	
Step Row Count			available
Final Row Count			available
Estimated Processing Time			available
Minimum Step Time			available
Join Type			available
Full Table Scan			available
multiple criteria joined by AND			available

Setting Classification for Throttles

Throttles limit the number of user queries that can run at the same time. Add classification settings to existing throttles or when creating a throttle. Classification options determine which queries match a rule.

When using the query characteristics classification criteria with throttles in Release 13.0 or earlier, the **Statement Type**, **AMP Limits**, and **Any Step Time Exceeds** criteria are available.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Throttles**.

3 In the **Throttles** tab, do one of the following:

- In **Name** or **Workload Name**, select a throttle.
- Click **Create Throttle**, enter a name and optional description (up to 80 characters long), select a rule type, and click **Save**.

4 Click the **Classification** tab.

5 In the **Add Classification Criteria** list, select **Request Source**, **Target**, **Query Characteristics**, **Query Band**, or **Utility**.

6 Click **Add**.

7 Specify options based on the classification criteria you selected.

8 If you selected the **Request Source** or **Target** criteria, do at least one of the following:

- In **Match String**, enter a string. A match string can contain ? to match exactly one character or an * to match zero or more characters. Use the **Match String Include** and **Exclude** buttons to add the match string to a list.
- Select **Items** from the list and use the **Items Include** and **Exclude** buttons to create your classification criteria.

9 Click **OK**.

Setting State-Specific Values for Throttles

Set state-specific query limits for existing system throttles or when creating a throttle.

You can override the default by setting query limits on a per-state basis. For example, you may want to raise the query limit of a system throttle during a low-traffic state, and lower the query limit during a high-traffic state.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Throttles**.

The **Throttles** tab appears, listing all existing throttles and their attributes.

3 Do one of the following:

- Select an existing system throttle from the **System Throttles** list.
- Select **Create Throttle**, enter a name and optional description (up to 80 characters long), select a rule type, and click **Save**.

4 Select **State Specific Settings**.

5 In **When Query Limit Exceeded**, select **Delay** or **Reject**. (Release 12.0.)

6 Click a state in the state matrix to set the throttle query limit for that state.

7 In **Edit [state name] Settings**, select **Create State Specific Settings**.

Note: If you select the **Use Default Settings** option, the settings at the bottom of the **Throttles** tab apply to the state.

8 Do one of the following:

- Next to **Query Limit**, select **Unlimited** to place no limits on the number of queries for this throttle.
- Enter a query limit number in the box, and select either **Delay** or **Reject**. (The **Delay** and **Reject** options are available in Release 13.0 or later.)
- Next to **Session Limit**, select **Unlimited** or enter a session-limit number in the box. (Release 12.0 and Release 13.0.)

9 Click **OK**.

Your selection is applied to each cell having that state, and overrides the setting specified in **Default Settings**.

10 [Optional] Change the **Default Settings** by selecting **Unlimited**, or enter a query limit number in the box and select **Delay** or **Reject**. (The **Delay** and **Reject** options are available in Release 13.0 or later.)

11 [Optional] Change the **Session Limit** by selecting **Unlimited** or entering a session limit number in the box. (Release 12.0 and Release 13.0.)

12 Click **Save**.

About Ruleset Workloads

A ruleset *workload* is a group of queries that share characteristics so that a set of workload management controls can be applied to the group. A workload has working values and defining characteristics that are evaluated during the classification phase of system management. High-quality workload management can improve response times and ensure more consistent response times for critical work.

The Workloads tab lists workload names, types, throttle limits, and enabled status. For each workload, you can specify one or more of the following:

- Classification criteria which determine if a query is assigned to the workload.
- Throttles which limit the number of concurrent active queries that can run.
- Service level goals which specify a goal for workload query performance.
- Exceptions which monitor queries and take specified actions if a query exceeds exception criteria while executing.

You can create up to 250 workloads. The following requests are examples of workloads:

- Batch jobs further subdivided by region or organization for reporting.
- Weekly or monthly reports that follow the calendar or a regular schedule.
- Jobs that are always critical whenever they occur.

Queries not classified into a specific workload are placed into the default workload named WD-Default. The WD-Default workload cannot be deleted or disabled.

Creating a Workload

You can group queries that share characteristics into a workload so that a set of workload management controls can be applied to the group.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Workloads**.

3 Click **Create Workload**.

4 In the **General** tab, enter a name.

5 [Optional] Enter a description up to 80 characters long.

6 Select an **Enforcement Priority** (the type of queries you expect to run in this workload):

- **Tactical** queries are short and high-priority.
- **Priority** queries are longer-running than tactical queries and higher priority.
- **Normal** queries are the customary queries running on the system.
- **Background** queries run when the system is not busy.

7 Click **Save**.

8 [Optional] Click the **Classification** tab.

9 [Optional] Determine if you want incoming queries classified into this workload. For example, select **Target** to have queries from a specific database sent to this workload.

10 [Optional] Click the **Throttles** tab.

11 [Optional] Determine if you want to set a throttle just for this workload. Any query that is classified into this workload is subject to this throttle.

12 [Optional] Click the **Service Level Goals** tab, and do one of the following:

- Select **Response Time Goal**, and enter a time in seconds for how quickly you want the queries to run. For example, type 4 if you want all queries to finish within 4 seconds. You can also set a service percentage. For example, type 90 if you expect 90% of queries to meet the time goal.
- Select **Throughput Goal**, and type a throughput number per hour. For example, type 200 if you expect 200 queries to be processed per hour.
- Select **No Goal** if you want no service-level goal for this workload.

13 [Optional] Click the **Exceptions** tab. Determine if you want an exception triggered for this workload. For example, if a query in this workload is taking too long, create an exception to move the query to a different workload.

14 Click **Save**.

Setting Classification for Workloads

Classification options determine which queries go into a workload. Add classification settings to existing workloads or when creating a workload. Classification options cannot be added to the default workload, WD-Default.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Workloads**.

3 Do one of the following:

- In **Name**, select an existing workload.
- Click **Create Workload**, enter a name and optional description (up to 80 characters long), select an **Enforcement Priority**, and click **Save**.

4 Click the **Classification** tab.

5 Select classification criteria from the list.

The classification types are **Request Source**, **Target**, **Query Characteristics**, **Query Band**, and **Utility**.

6 Click **Add**.

7 Specify options based on the classification criteria you selected.

8 If you selected **Request Source** or **Target**, do at least one of the following:

- In **Match String**, enter a string. A match string can contain ? to match exactly one character or an * to match zero or more characters. Use the **Match String Include** and **Exclude** buttons to add the match string to a list.

- Select **Items** from the list and use the **Items Include** and **Exclude** buttons to create your classification criteria.

9 Click **OK**.

Setting State-Specific Throttles for Workloads

Set state-specific throttle query limits for existing workloads or when creating a workload. If you select the **Use Default Settings** option, the settings at the bottom of the Throttles tab apply to the state.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Workloads**.

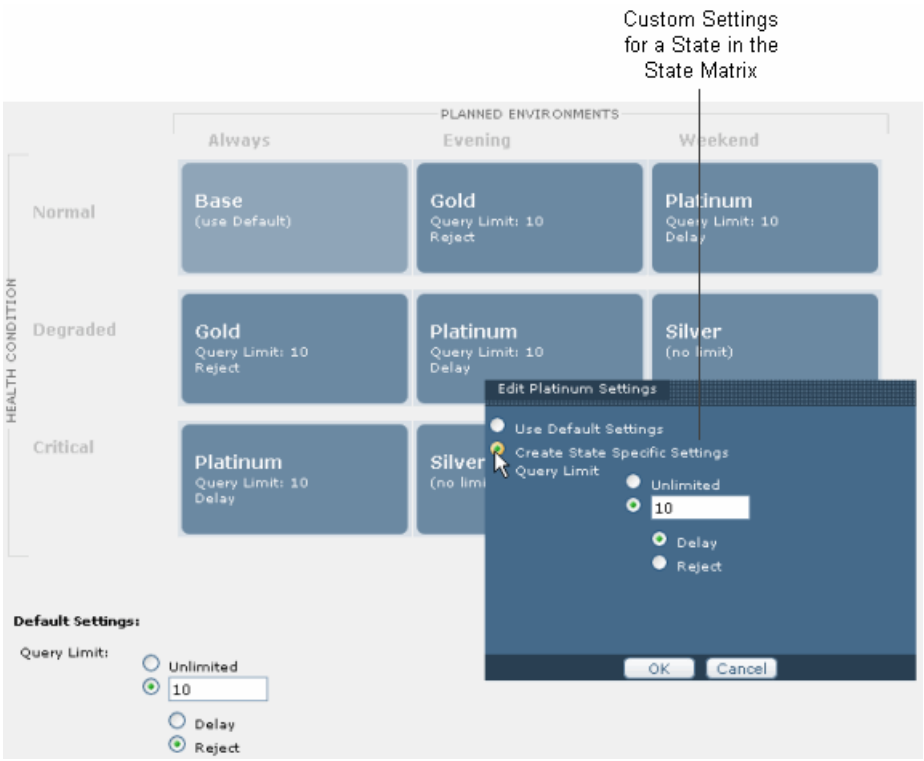
3 Do one of the following:

- In **Name**, select an existing workload.
- Click **Create Workload**, enter a name and optional description (up to 80 characters long), select an **Enforcement Priority**, and click **Save**.

4 Click the **Throttles** tab.

5 Click on a state in the state matrix.

6 In **Edit [state name] Settings**, select **Create State Specific Settings**.



7 Do one of the following:

- Select **Unlimited** to prevent limits on the number of queries for this workload.

- Enter a query limit number in the box, and select either **Delay** or **Reject**. (The **Delay** option is available in Release 13.0 and later.)

8 Click **OK**.

Your selection is applied to each cell having that state and overrides the setting specified in **Default Settings**.

9 [Optional] Change the default setting by selecting **Unlimited** or entering a number in the box so queries over the limit are rejected. (In Release 13.0 and later, **Delay** and **Reject** options are available.)

10 Click **Save**.

Setting Service-Level Goals

Service level goals (SLGs) help determine if workload management is meeting expectations. You can state service-level goals in throughput or response time with a service percentage.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Workloads**.

3 Do one of the following:

- In **Name**, select an existing workload.
- Click **Create Workload**. Enter a name, optional description (up to 80 characters long), and enforcement priority. Click **Save**.

4 Click the **Service-Level Goals** tab.

5 Do one of the following:

- Select **Response Time Goal**, enter a time in seconds, and set a service percentage. For example, if you want 90% of queries to finish within 4 seconds, set a response time of 4 and a service percent of 90. You can have different goals in different planned environments. For example, you can set a response time of 4 seconds for a daytime environment and 12 seconds for a nighttime environment. Metrics graphs show response times.
- Select **Throughput Goal** and enter a throughput number per hour.
- Select **No Goal** if you want no service-level goal. This option may be appropriate, for example, for very low priority background workloads.

6 Click **Save**.

Defining a Workload Exception

A *workload exception* is one or more events that, when all occur at the same time (or continuously for a specified period of time) cause a defined action to occur. You can add multiple criteria to an exception. If you add more than one criteria for an exception, all criteria must be satisfied for the exception to be triggered.

Note: Some exception criteria give you the option to add qualification time. Qualification time is the amount of time the criteria must take place to be recognized. Setting qualification time prevents very short incidents of a criteria from being acknowledged.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Workloads**.
3. Do one of the following:
   - In **Name**, select an existing workload.
   - Click **Create Workload**, enter a name, optional description (up to 80 characters long), and enforcement priority, and click **Save**.
4. Click the **Exceptions** tab.

   Note: In Release 12.0 and Release 13.0, local exceptions are displayed separately from shared exceptions on the **Exceptions** tab.
5. Click **Create Exception**.
6. Enter a name.
7. [Optional] Enter a description up to 80 characters long.
8. In **Criteria**, select an option from the list.
9. Click **Add**.
10. Specify the additional required information.
11. In **Actions**, select one of the following actions to be performed when the specified criteria occur:
    - Select **Notification Only** to send notification, and take no other action.
    - Select **Abort** to stop the query.
    - Select **Abort Selects Only** to stop the query only if it is a SELECT.
    - Select **Change Workload to** to change the query workload to the workload you select from the list.
12. [Optional] In **Notifications**, select any of the following options:
    - Select **Alert** to specify the Teradata Alerts to trigger at the start of the event. Select an alert name from the list.
    - Select **Run Program** to specify the Teradata Alerts registered programs to trigger at the start of the event. Select a program from the list.
    - Select **Post to QTable** to post the text you specify to the system queue table when the event starts. (This option is not integrated with Teradata Alerts.)
13. Click **OK**.
14. If there is more than one local exception assigned to the **Change Workload** action, use the **Exception Precedence** tab to set the priority of the exceptions. (Release 12.0 and Release 13.0.)

Viewing Workload Service Level Goal Summary

A service level goal for workload query performance is set in either throughput or response time with a service percentage. You can view a summary of service level goals that have been set for individual workloads.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Workloads**.
3. Click the **SLG Summary** tab.
   Workloads are listed, along with their service level goals for each planned environment.

About Evaluation Order

Evaluation order determines the placement of queries into workloads and utility jobs into rules. Setting evaluation order is useful when you have created several workloads or utility sessions.

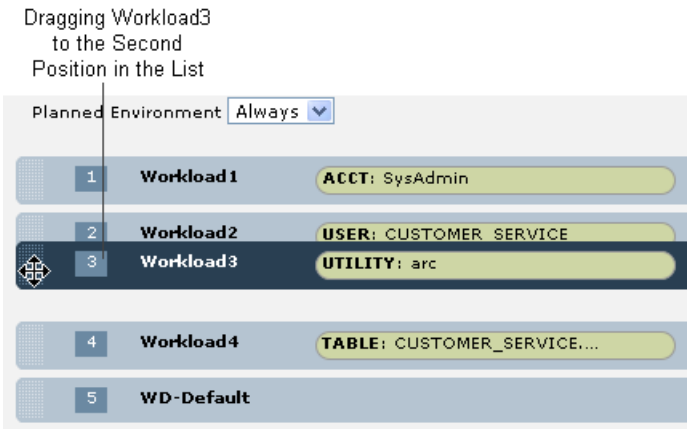
For workloads, if classification criteria can sort the same query into different workloads, evaluation order determines the workload in which the query is placed. For example, account finance users are classified into one workload and users in a specific geographical location are classified into a different workload. If an individual is a member of both groups, the workload highest on the evaluation order list is the workload in which the individual is sorted.

For utility sessions, if a utility job matches more than one utility session rule, evaluation order determines the rule in which the utility job is placed.

Setting Evaluation Order for Workloads

If classification criteria can sort the same query into different workloads, evaluation order determines the workload in which the query is placed. The workload in the highest position on the Evaluation Order tab is applied.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Workloads**.
3. Click the **Evaluation Order** tab.
4. Select a **Planned Environment** from the list.



5. In the list of workloads, do one of the following to order the list and place higher priority workloads at the top:

- Drag workload names to reorder.
- Click in a number box, highlight the existing number, and enter a new number.

6 Click **Save**.

Setting Enforcement Priority for Workloads

Enforcement priority establishes the type of queries you expect to run in a specific workload.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Workloads**.

3 Do one of the following in the Workloads tab:

- In **Name**, select an existing workload.
- Click **Create Workload**, and enter a name.

4 Select an **Enforcement Priority**:

- **Tactical** queries are short and high-priority.
- **Priority** queries are longer-running than tactical queries and high-priority.
- **Normal** queries are the regular queries running on the system.
- **Background** queries run when the system is not busy.

5 Click **Save**.

Adjusting Priority Distribution for Workloads

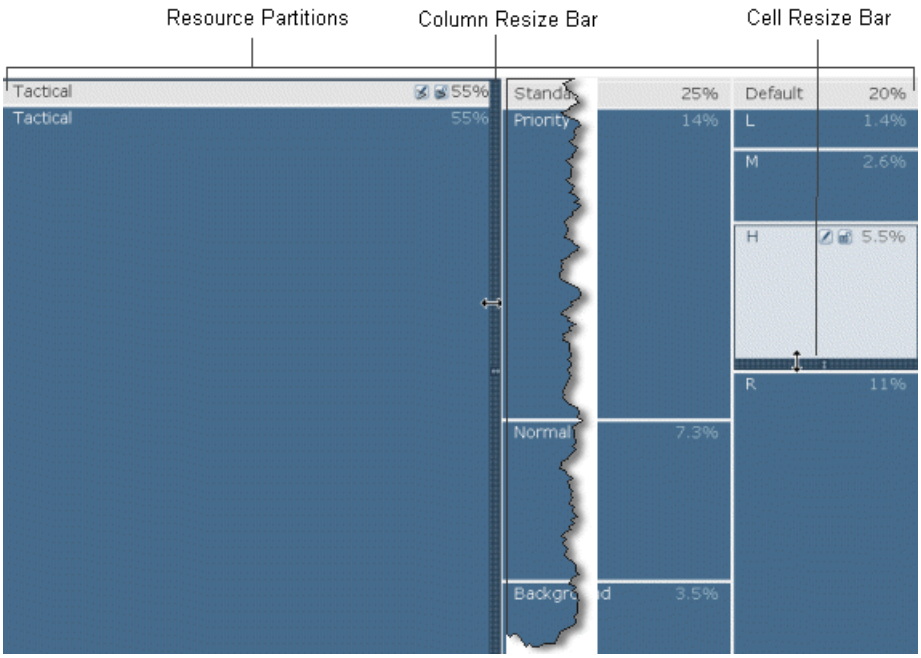
On the Priority Distribution tab, each cell in the diagram is an allocation group. The size of each cell is proportional to the relative weight. Each column in the diagram is a resource partition. The width of each column is proportional to the relative weight of the resource partition. Drag cell borders to dynamically change the amount of CPU (as a percentage) allocation groups are given. Drag column borders to change the amount of CPU available to resource partitions.

Note: If you modify the base state on the Priority Distribution tab, the changes are applied to other states as long as no change has been made to any of the other states. Once any of the states are changed, the only values that are applied are the changes to system values.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Workloads**.

3 Click the **Priority Distribution** tab.



4 In **State**, select a state from the list.

5 For resource partitions, do any of the following:

- Mouse over any resource partition name to obtain information about the partition.
- Click on a resource partition name, and drag the column resize bar. This resizes the column and reallocates space between the resource partitions.
- Click on a resource partition name, and click . Enter a system percentage allocation and, optionally, a CPU percentage limit. If you select the **Locked** option, this column becomes fixed in the diagram, and other columns can be dynamically resized around this column.
- Click on a resource partition name and click or to unlock or lock the column.

6 For allocation groups, do any of the following:

- Mouse over any allocation group to obtain information about the group.
- Click in a cell, and drag the cell resize bar. This resizes the cell and reallocates space within a resource partition.
- Click in a cell, and click . Enter a system percentage allocation and, optionally, a CPU percentage limit. If you select the **Locked** option, this cell becomes fixed in the diagram and other cells can be dynamically resized around this cell.
- Click in a cell and click or to unlock or lock the cell.

7 [Optional] Select **System Values**, and enter a CPU limit to set the maximum percentage of system CPU that Teradata Database can use.

8 [Optional] Select **Reserved AWTs** (AMP Worker Task), and enter a number to set the maximum number of AWTs that are reserved for tactical workloads.

9. [Optional] Select **Max Expedited AWTs**, and enter a number to set the maximum number of AWTs that are created on each node.
10. Click **Save**.

Mapping Console Utilities to Workloads

Map console utilities to workloads so utilities can be organized and prioritized.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Workloads**.
3. Click the **Console Utilities** tab.
4. In **Console Utility to Workload Mapping**, use the lists to set a workload name for each console utility.

   The lists contain the predefined workload, WD-Default, and any workloads you have created. WD-Default is used when no other workload is specified.
5. Click **Save**.

About Performance Groups

A *performance group* is a collection of parameters used to control and prioritize resource allocation for a particular set of database sessions. Performance groups belong to a resource partition, a set of prioritized performance groups related by their users' associations. Resource partitions have an assigned weight that determines the proportion of resources available to that partition relative to other partitions.

On the **Console Utilities** tab, the mapping of performance groups to workloads only applies to console utilities that are not included in the **Console Utility to Workload Mapping** section.

Creating a Performance Group

Create a performance group to determine which allocation group manages a query. Performance groups belong to a resource partition, a collection of prioritized performance groups related by their users' associations.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Workloads**.
3. Click the **Console Utilities** tab.
4. Click **Create Performance Group Mapping**.
5. Mouse over **New Performance Group**, and click .
6. Enter a name.
7. Click outside the name.
8. In **Workload Name**, select a workload from the list for the performance group.
9. Click **Save**.
10. [Optional] To delete a performance group, mouse over the group name, and click .

Mapping Performance Groups to Workloads

Map performance groups to workloads so the groups can be prioritized along with other work.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Workloads**.
3. Click the **Console Utilities** tab.
4. In **Performance Group to Workload Mapping**, use the lists to set a **Workload Name** for each **Performance Group**.

   The lists contain the predefined workload, WD-Default, and any workloads you have created. WD-Default is used when no other workload is specified.
5. Click **Save**.

About Allocation Groups

An *allocation group* can limit the total amount of CPU used by sessions under its control. An allocation group has an assigned weight that is compared to other allocation group weights. The four default allocation groups are tactical, priority, normal, and background. The default allocation groups and any groups you create are assigned to resource partitions.

Note: The allocation groups named L, M, H, and R stand for Low, Medium, High, and Rush. Do not use or modify the allocation groups named L, M, H, and R. The groups represent levels of priority and are used by the system for console utilities.

By default, individual workloads are associated with a specific allocation group based on the enforcement priority assigned to the workload. You can change the default mapping using the diagram on the **Workload Mapping** tab. The diagram enables you to reassign workloads to allocation groups and reassign allocation groups to resource partitions. Allocation groups that contain workloads cannot be deleted.

Carefully consider the allocation groups you drag in and out of the tactical resource partition because this can have a significant effect on the amount of CPU available to the queries running in the allocation group. The tactical resource partition normally has a much higher weight than other resource partitions. If you move an allocation group containing long, resource-intensive queries into the tactical resource partition, those queries can consume a large amount of the available CPU, impacting queries running in other resource partitions. If you move a tactical allocation group out of the tactical resource partition, the queries running in the tactical allocation group are assigned a lower priority and may not meet their response-time goals.

Creating an Allocation Group

Create allocation groups using the Workload Mapping tab.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Workloads**.
3. Click the **Workload Mapping** tab.
4. Mouse over the name of the resource partition, such as **Tactical** or **Standard**, to which you want to add an allocation group, and click  .

5 Enter a name.

6 Click outside the name.

7 [Optional] To move the allocation group to a different resource partition, mouse over the name of the group, and drag the group to a different column.

8 [Optional] To change the name of the allocation group, mouse over the name of the group, and click .

9 Click **Save**.

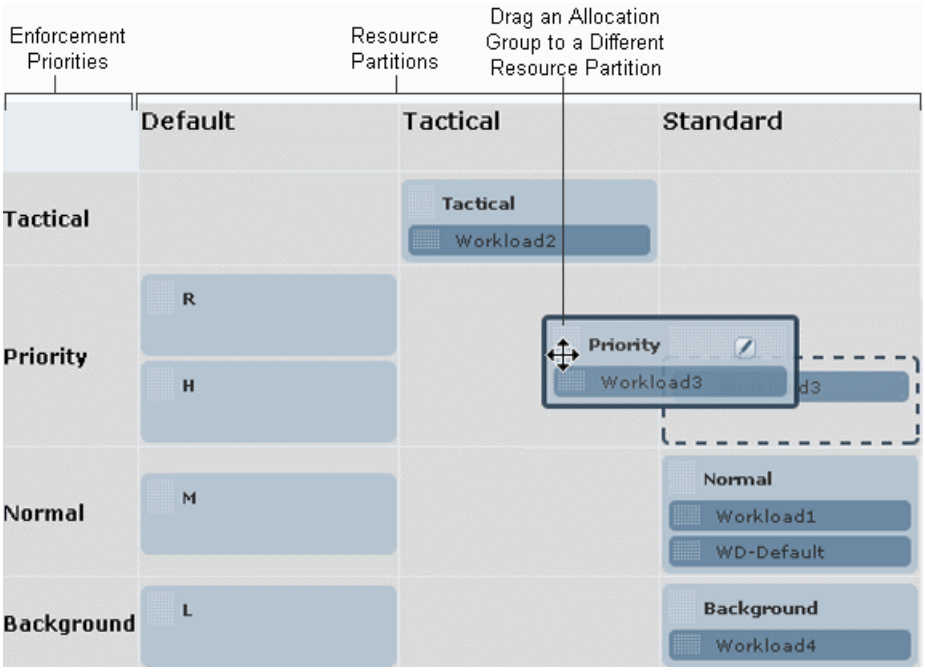
Mapping Allocation Groups to Workloads

If you have created workloads, you can view the **Workload Mapping** tab to see how the workloads map to allocation groups and resource partitions. Allocation groups and resource partitions determine how much CPU is available to the queries that are running in the listed workloads. The allocation groups named L (Low), M (Medium), H (High), R (Rush) represent levels of priority. These groups are used by the system for console utilities and cannot be deleted.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Workloads**.

3 Click the **Workload Mapping** tab.



4 Do any of the following:

- Create allocation groups by mousing over a resource partition name, such as Tactical or Standard, and clicking .
- Create a resource partition by mousing over **Resource Partitions** and clicking .

- Drag an allocation group to a different resource partition.
- Drag a workload to a different allocation group of same priority in any resource partition.
- Rename a resource partition by mousing over a name and clicking .
- Delete an empty allocation group by mousing over a name and clicking .
- Delete an empty resource partition by mousing over a name and clicking .

5 Click **Save**.

Deleting An Allocation Group

You cannot delete the L (Low), M (Medium), H (High), R (Rush) allocation groups or allocation groups that contain workloads.

1 Edit or create a ruleset.

2 From the ruleset toolbar, click **Workloads**.

3 Click the **Workload Mapping** tab.

4 Mouse over the name of an allocation group, and click .

5 Click **Save**.

About Ruleset Exceptions

An *exception* is a collection of one or more events that, when all occur at the same time (or continuously for a specified period of time), cause a defined action to occur. When creating or editing a ruleset, you can specify a set of exception criteria and exception actions to apply to workloads for each planned environment.

The tabs on the ruleset **Exceptions** view display all exception information for a ruleset from different perspectives:

By Planned Environment

Exceptions organized by planned environment. After selecting a planned environment, all exceptions are listed alphabetically by name. All workloads are listed in workload evaluation order. Select and clear the workload check boxes to enable or disable the use of individual exceptions for individual workloads. If the All-WD check box is selected, all check boxes in the row are selected and cannot be individually controlled.

By Workload

Exceptions organized by workloads. After selecting a workload, all exceptions are listed by name. All planned environments are listed by name and by how they are arranged in the state matrix. Select and clear the planned environment check boxes to enable or disable individual exceptions for individual planned environments. Within a workload, if an exception is enabled for All-WD in a specific planned environment, the check box is selected and disabled. (You can change the exception setting on the **Planned Environment** tab.) In Release 12.0 and Release 13.0 local exceptions are displayed separately from shared exceptions.

By Exception

List of all exceptions created for planned environments and workloads. After selecting an exception, all workloads are listed in workload evaluation order. All planned environments are listed by name. Select and clear the planned environment check boxes to enable or disable individual workloads for individual planned environments. If the All-WD check box is selected, all check boxes in the column are selected and cannot be individually controlled.

Defining an Exception

An *exception* is defined by a collection of one or more threshold events that, when all occur at the same time and for a specified period of time, cause a defined action to be taken. If you add more than one criteria, all criteria must be satisfied in order for the exception to be triggered. Only one qualification time can be specified even if multiple qualified criteria are added. If qualified criteria are specified, the exception is triggered only if all of the qualified criteria are satisfied for the duration of the qualification time. After an exception is created, additional controls in the tabs of the **Exceptions** view allow you to manage the exception.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Exceptions**.
3. From any of the tabs in the **Exception** view, click **Create Exception**.
4. Enter a name.
5. [Optional] Enter a description up to 80 characters long.
6. Select an option from the **Exception Criteria** list, and click **Add**.
7. Enter a value for the criteria.
8. For qualified criteria (marked with *), enter a value in **Qualification Time**.
9. Select one of the following actions to be performed when the specified criteria are satisfied:
   - **Notification Only** sends a notification only. No other action is performed.
   - **Abort** stops the query.
   - **Abort Selects Only** stops the query only if it is a SELECT.
   - **Change Workload to** changes the workload of the query to the specified workload.

   Change Workload to is unavailable if the **Blocked Time** criterion or **Elapsed Time** criterion was selected.
10. [Optional] Select any of the following notifications:
    - **Alert** sends the selected alert.
    - **Run Program** starts the selected program.
    - **Post To QTable** posts the string entered in the box to the QTable. (In Release 12.0, you can select the **Post to QTable** option, but the text box is not available.)
11. Click **OK**.
12. If there is more than one shared exception assigned the **Change Workload to** action, use the **Exception Precedence** tab to set the priority of the exceptions.

(Release 12.0 and Release 13.0.)

About Tactical CPU per Node Exceptions

If you select the **Tactical CPU per Node** exception criteria, the following rules apply:

- The exception can be enabled for workloads that use the tactical enforcement priority. Tactical workloads are in the top row of the **Workload Mapping** tab.
- The exception must have a **Change Workload to** action to another workload in the same resource partition column.
- The exception must have **Tactical CPU per Node** set to five cpu-seconds or less.
- The exception must have **CPU Time** set to a number greater than **Tactical CPU per Node**.
- Using **Tactical CPU per Node** criteria disables the **Blocked Time** and **Elapsed Time** criteria for the exception.

Managing Exceptions

Ruleset exceptions are managed from the ruleset **Exceptions** view. In Release 12.0 and Release 13.0, local exceptions are displayed separately from shared exceptions on the **by Workload** tab.

1. Edit or create a ruleset.
2. From the ruleset toolbar, click **Exceptions**.
3. [Optional] Create exceptions by clicking **Create Exception** from any tab.
4. After exceptions have been created, click the **by Planned Environment**, **by Workload**, or **by Exception** tab to manage exceptions.
5. Depending on the tab, select **Workload** or **Exception** from the list.
6. [Optional] Edit an exception from the **by Planned Environment** or **by Workload** tab by mousing over the exception and clicking  .
7. [Optional] Delete an exception from the **by Planned Environment** or **by Workload** tab by mousing over the exception and clicking  .
8. In the selected view, select or clear the appropriate check boxes to enable or disable the exceptions for the listed workloads, planned environments, or workload and planned environment combinations.
9. Click **Save**.