Using the Cisco Modeling Labs Client
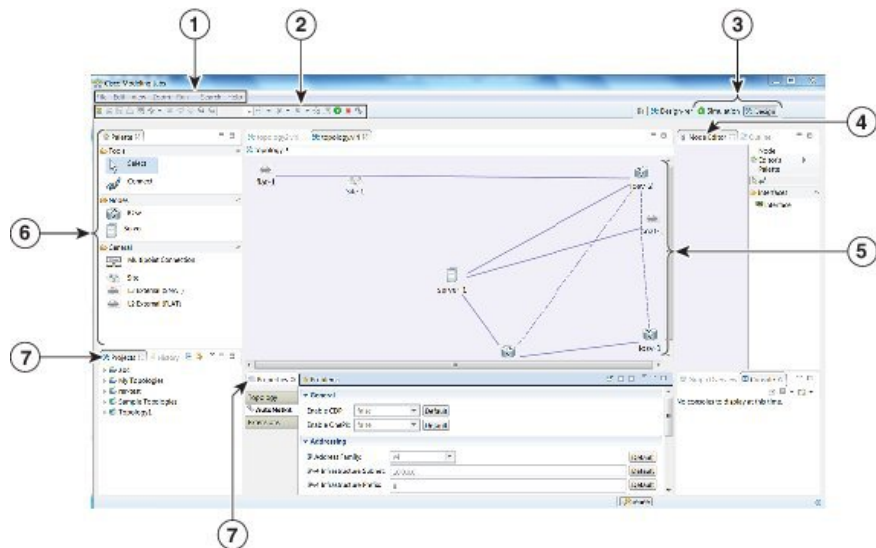
Using the Cisco Modeling Labs Client Overview

The Cisco Modeling Labs client is a cross-platform, point-and-click GUI that simplifies topology creation, initial-device configurations, and permits access to the Cisco Modeling Labs server. You can interact directly with your running simulations from this GUI. Additionally, the Cisco Modeling Labs client provides the functionality to generate default router configurations before simulating your topology.

This chapter introduces the main areas and capabilities of the Cisco Modeling Labs client in the following section.

2

Navigating Within the Cisco Modeling Labs Client

This section describes the functionality of the Cisco Modeling Labs client, which comprises of a workbench containing a menu bar, a toolbar, multiple editors, multiple perspectives, and multiple views.

Figure 1: Workbench Components

Callout	Description	Callout	Description
1	Menu Bar	5	Canvas
2	Toolbar	6	Palette
3	Perspectives	7	Views
4	Node Editor	5+6 combined	Topology Editor

- **Workbench**—Refers to the Cisco Modeling Labs client desktop environment. Each time the workbench is exited, it is automatically saved, including all the open perspectives, views, and editors. When the workbench is reopened, it appears exactly as it was when last closed.

- **Menu Bar**—References all the actions that can be performed when using the Cisco Modeling Labs client.

- **Toolbar**—Contains a set of icons representing commands. The Toolbar provides shortcuts to actions that are used most often from the Menu bar.

- **Cisco Modeling Labs Client Perspectives**—Identifies the **Design** and **Simulate** perspectives, each of which is associated with an initial set of views and editors in your workbench.

◦ From the **Design** perspective, you can design your network topology, build the node configurations, and check routing protocols. The **Design** perspective is the default perspective if you are launching the Cisco Modeling Labs client for the first time.

◦ From the **Simulate** perspective, you can enable devices and modify configurations to run the simulations. When the nodes in your topology are fully initialized, you can connect to the consoles as you would connect to a router console.

- **Cisco Modeling Labs Client Editors**—Provides alternative components within the Cisco Modeling Labs client from which you can create and edit topologies. Two editors are provided: **Node Editor** and **Topology Editor**.

  ◦ The **Node Editor** shows the interface details for selected elements in the **Topology Editor**.

  ◦ The **Topology Editor** is comprised of the **Palette** view and the canvas.

- **Cisco Modeling Labs Client Views**—Provides alternative presentations of your topology, and methods for navigating the information in your workbench.

- **Cisco Modeling Labs Client Layout**—Enables you to personalize your workbench, allowing you to rearrange, resize, reset, and move between views.

Menu Bar

The menu bar provides access to the complete list of actions that are possible in the Cisco Modeling Labs client.

Table 1: Menu Bar Items

Menu	Action(s)
File	Enables you to perform actions to a topology project, sub topology, or topology .virl file; set preferences for resources; and exit the Cisco Modeling Labs client. • **New**—Creates a new topology project folder, topology project, or topology .virl file. See Design a Topology Overview for more information. • **Save**—Saves the current Topology Editor contents. • **Save As**—Saves the current Topology Editor contents to a new file. • **Print**—Prints the current Topology Editor design. • **Import**—Imports a resource type (for example, a topology.virl file) into the Cisco Modeling Labs client. See Importing a Topology File, on page 6 for more information. • **Export**—Exports a resource type (for example, a topology.virl file) from the Cisco Modeling Labs client. See Exporting a Topology File, on page 6 for more information. • **Close**—Closes the current editor. • **Close All**—Closes all open editors. • **Preferences**—Opens the **Preferences** dialog box where you can update the settings for the Cisco Modeling Labs client. See Setting Preferences for the Cisco Modeling Labs Client, on page 46 for more information. • **Switch Workspace**—Allows you to select a workspace folder to use for your session. • **Exit**—Exits the Cisco Modeling Labs client.

Menu	Action(s)
Edit	Enables you to manipulate resources on the canvas. • **Undo**—Cancels the most recent change. • **Redo**—Applies the most recent change that was removed. • **Copy**—Places a copy of the topology on the clipboard. • **Paste**—Pastes the topology from the clipboard to the canvas at the current cursor location. • **Delete**—Deletes the current topology. • **Select All**—Selects all objects in the current topology canvas. • **Show Map Background**—Displays a map background for the topology. (You must have Internet access to use this feature.) • **Reset Note Subtype**—Enables you to redefine a virtual node on the canvas. The node subtype installed with Cisco Modeling Labs is Cisco IOS. When **Reset all interface names** is selected, the interface IDs are reset sequentially, starting from the Minimum range. • **Highlight Connection**—Highlights all connections to and from the selected node. • **Distribute Nodes**—Aligns the nodes on the canvas evenly, either vertically or horizontally, or places them in a grid layout. (This feature is disabled when **Show Map Background** is chosen.) • **Layout Nodes**—Places nodes on the canvas in a tree layout or an F-R layout. (This feature is disabled when **Show Map Background** is selected.) • **Alignment**—Aligns the nodes on the canvas. Options are: left, center, right, top, middle, or bottom. (This feature is disabled when **Show Map Background** is selected.) • **Group to Site**—Groups two or more nodes within a site. **Note** Sites are created within Cariden imports. • **Ungroup to Site**—Ungroups nodes from within a site, removing one layer of nesting.
View	Enables you to see numerous perspectives within the Cisco Modeling Labs client. • Lists the view options for the selected perspective within the Cisco Modeling Labs client. • Allows you to filter your view results.
Zoom	Provides zoom in and zoom out functionality from within the canvas.
Run	Enables you to start and stop simulations, and to generate node configurations.
Search	Enables you to perform topology searches and text searches within workspaces, topology projects, files, and working sets.

Menu	Action(s)
Help	Displays the help topics for using the Cisco Modeling Labs client in a separate browser window. • **Help Contents**—Opens the Cisco Modeling Labs client support documentation. • **Cheat Sheets**—Helps you complete a task. • **Generate Problem Report**—Generates a detailed report of a problem, which can be submitted to Cisco Technical Assistance Center (Cisco TAC) for further investigation. See Generating Problem Reports, on page 7 for more information. • **About Cisco Modeling Labs**—Identifies version information and displays details pertaining to software installations.

Importing a Topology File

Step 1   From the menu bar, choose **File** > **Import**.
The **Import** dialog box appears.

Step 2   From the drop-down list, choose **Topology** > **Import Topology file from File System**, and click **Next**.

Step 3   Click **Browse** to locate the applicable .virl file.

Step 4   In the right pane, check the check box for the applicable .virl file.

Step 5   Click **Finish** to import the topology file.

Exporting a Topology File

Step 1   From the menu bar, choose **File** > **Export**.

     Note   To export a topology file, it must be open on the canvas of the **Topology Editor**.

     The **Export** dialog box appears.

Step 2   From the drop-down list, choose **Topology** > **Export Topology file to File System**, and click **Next**.

Step 3   Click **Browse** to select the directory to export the topology file to.

Step 4   Click **Finish** to export the topology file.

Generating Problem Reports

The Cisco Modeling Labs client provides functionality that allows you to generate problem reports for any problems encountered in your topology. It is accessible from the menu bar under **Help** > **Generate Problem Report**.

While all options are pre-selected, you can individually select the information you want to include in the report.

Note When generating a problem report for your topology, you must have the topology containing the problem open on the canvas.

Option	Description
Log File	User interface .log file from the user's workspace.
Consoles Content	Current content from the **Console** view messages. These are messages from the server in response to AutoNetkit and simulation launch actions.
Topology Editor Contents	Contents of the currently open topology file in the **Topology Editor** canvas.
Screenshot of the User Interface	Screenshot showing the state of the user interface when the problem occurred.
Web Services Setting	Report of the Web Services details and errors.
Additional Information	Any additional information that users can provide to describe the problem.

The generated problem report is saved to a .zip file where you can check the contents before sending it to the Cisco Technical Assistance Center (Cisco TAC) for investigation.

Toolbar

The toolbar is a compilation of icons representing commonly used actions. The toolbar is arranged below the menu bar and offers the same actions as the menu bar, in a single click. The following table outlines the actions that can be performed using the Cisco Modeling Labs client toolbar.

Icon	Function	Description
	New Topology Project	Creates a new topology project. A topology project is a folder in which multiple topology files are stored. If you create many different topologies, you can set up different projects for the topologies you create.
	New Topology File	Creates a new topology file. A topology file is a .virl file where the network arrangement is designed.

Icon	Function	Description
	Save	Saves the current topology.
	Print	Prints the current topology.
	Show Map Background	Uses Global Positioning Satellite (GPS) coordinates to map a topology to precise geographic locations. **Note** You must have Internet access to use this feature since it uses the map service from MapQuest to retrieve the latest map images.
	Reset Node Subtype	Resets the node subtypes. A node subtype defines the virtual machine running a particular operating system, for example, Cisco IOSv virtual software. You can also choose to reset all interface names.
	Delete	Deletes the currently selected element within the topology.
	Undo	Undoes the most recent action.
	Redo	Redoes the most recent undone action.
	Zoom In	Enlarges the topology view on the canvas.
	Zoom Out	Decreases the topology view on the canvas.
	Distribute Nodes	Aligns the nodes on the canvas either vertically or horizontally. Alternatively, arranges the nodes in a grid.

Icon	Function	Description
	Layout Nodes	Places the nodes on the canvas in a tree layout or an Fruchterman-Reingold (F-R) layout. • A basic tree layout allows you to choose a node as the root of the tree, and the nodes connected to the root become the children of that root node. • An F-R layout has connected nodes close to each other, while unconnected nodes are further apart from each other.
	Alignment	Aligns the nodes on the canvas either horizontally or vertically, and allows you to position them at various points on the canvas.
	Group to Site	Groups two or more nodes within a site. Sites can be nested, forming a hierarchical structure.
	Ungroup Site	Ungroups nodes from within a site, removing one layer of nesting.
	Launch Simulation	Launches a simulation.
	Stop Simulations	Stops a running simulation.
	Update Router Configurations	Generates configurations for the topology.

Cisco Modeling Labs Client Components

The three main components of the Cisco Modeling Labs client are described in the following sections:

- Cisco Modeling Labs Client Editors, on page 10

- Cisco Modeling Labs Client Perspectives, on page 11

- Cisco Modeling Labs Client Views, on page 15

Cisco Modeling Labs Client Editors

Editors are visual components within the Cisco Modeling Labs client. Currently, two editors are available:

- **Topology Editor**—Shows the entire topology (or sites). The Topology Editor comprises the canvas and the Palette.

- **Node Editor**—Shows the interface details for the currently selected elements in the Topology Editor.

Note   Selecting a connection in the Topology Editor shows the details of both the endpoints in the Node Editor.

Topology Editor

The Topology Editor allows you to:

- Add, move, group, rename, and delete nodes and sites on the canvas, or change the properties.

- Create or remove connections between nodes.

- Use GPS coordinates to map your topology to precise geographic locations.
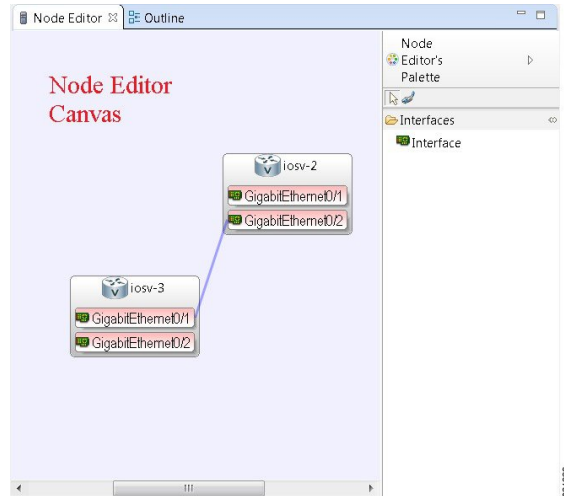
Figure 2: Topology Editor

Node Editor

The Node Editor allows you to:

- Add interfaces to a node.

- Add connections to the interfaces on a node.

- View nodes, their connections, and all the interfaces on the nodes.

- Specify which interface the connection connects to on each node.

• Update properties for interfaces.

Figure 3: Node Editor

Cisco Modeling Labs Client Perspectives

A perspective defines the initial set and layout of views and editors in the workbench. The Cisco Modeling Labs client provides two perspectives. However, you can customize your own user-defined perspectives for use, which can be saved or deleted, as needed.

The two perspectives provided in the Cisco Modeling Labs client are: **Design** and **Simulation**:

- **Design**—Allows you to create and design your topologies, for example, adding devices and defining interfaces, and adding connections to devices within your network. If you are using the Cisco Modeling Labs client for the first time, the **Design** perspective opens by default.

- **Simulation**—Allows you to simulate running configurations that are generated from virtual Cisco IOS versions.

Note The **Design** and **Simulation** perspectives that are built into the Cisco Modeling Labs client cannot be deleted.

One or more perspectives can exist in a single workspace. If multiple perspectives are opened at the same time, you can choose whether to layer them, or open them in a separate workspace.

Working with Perspectives

You can manage the various perspectives defined in the workbench under Perspectives preferences, choosing **File** > **Preferences** > **General** > **Perspectives**.

Figure 4: Perspective Preferences
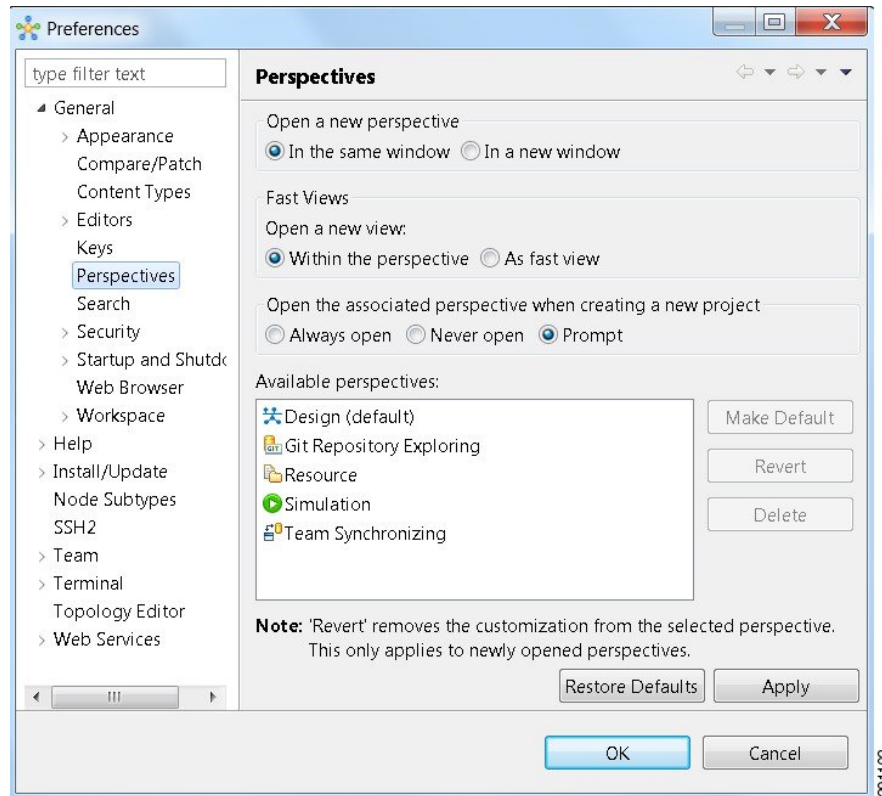
The following table describes the **Perspectives** options:

Table 2: Perspectives Options

Option	Description
Open a new perspective	Defines whether a new perspective opens in the current workbench, or opens in a new window. By default, a new perspective opens in the current workbench.
Open a new view	Defines whether a new view opens within the current perspective, or opens docked beside the current perspective (fast view). By default, a new view opens within the current perspective.
New project options	Defines perspective behavior when a new project is created. By default, a new project opens the perspective in the same workbench.

The following table describes the **Available Perspectives** options:

Table 3: Available Perspectives Options

| Option	Description
Make Default	Sets the selected perspective as the default perspective.
Revert	Resets the definition of the selected perspective to the default configuration. This option is only applicable to system-defined perspectives.
Delete	Deletes the selected perspective. This option is only applicable to user-defined perspectives. (System-defined perspectives cannot be deleted.)

Customizing Perspectives

Cisco Modeling Labs 1.0 provides two perspectives for use in the workspace: the **Design** perspective and the **Simulation** perspective. These perspectives display various views and settings that cannot be changed, nor can they be deleted from the Cisco Modeling Labs 1.0 client. You can, however, customize additional **Design** and **Simulation** perspectives for your specific needs.

From the **View** menu, you can open additional views and arrange them in your perspective. The views can be arranged by dragging them around the workspace. When finished, you can save the perspective.

Note Right-click the applicable perspective tab to open the context menu, and then click **Show Text**. The **Design** perspective and the **Simulation** perspective buttons will be displayed as text labels instead of icons only.

Step 1 Open the **Design** perspective or **Simulation** perspective in your workspace.
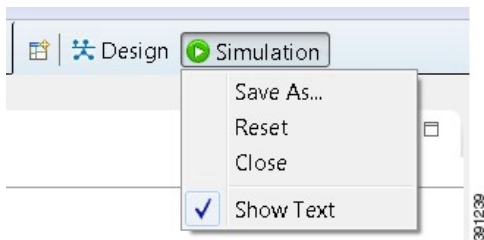
Use this as an initial template from which to create a customized perspective.

Step 2 To add new views, open the **View** menu to display the various views for use.
Views already in use are shown as dimmed in the list.

Step 3 To add node information, double-click a node to add a **Node Editor** in the perspective.

Step 4 Click and drag the views and **Node Editors** to different positions within the workspace. For example, you can move each node's **Console** view so they are either side-by-side, or stacked in tabs.

Step 5 When you are finished arranging the workspace, right-click the applicable perspective button, and then click **Save As**. You are prompted to name the new perspective.

Step 6   Enter a name for the perspective and click **OK**.

Design Perspective

The **Design** perspective allows you to create and design your topologies. By default, the **Design** perspective incorporates the components listed here because they are the most widely used. However, you can customize a Design perspective to include a different set of components.

- **Palette** view—Provides the node types, connection types, and sites used to design a topology.

- **Projects** view—Lists topology projects, subfolders, and files defined from the workbench.

- **History** view—Lists changes made to a file based on date and time stamp.

- **Properties** view—Identifies node and interface properties.

- **Outline** view—Lists structural elements of a topology.

- **Node Editor**—Identifies nodes, node connections, and node interfaces.

- **Topology Editor**—Develops a network topology.

- **Graph Overview**—Provides methods for viewing a network topology.

To customize a new **Design** perspective, choose the desired components from the **View** menu (for example, **View** > **Other** > **Cisco Terminal**), and then drag that component view to the desired location within the workbench. When you are done adding components to the workbench and the component views are laid out as desired, select **Save As** from the **Design** toolbar and enter a name for the new **Design** perspective.

From the **Design** perspective, right-click the Design icon. This displays the following menu options:

Table 4: Design Perspective Context Menu Options

Operation	Description
Save As	Saves the customized workbench layout, views, and editors as a new Design perspective.
Reset	Resets the current perspective to display the workbench design that was used when the workbench was first opened. **Note** Any open editors, for example, Node Editor, are not closed when a perspective is reset. You must close them manually.
Close	Closes the current perspective. You can reopen it using the **Open Perspective** tool, which is located at the upper right corner of the window.
Show Text	Shows perspective names in the toolbar instead of icons, when selected. When deselected, shows perspective icons in the toolbar instead of names.

Simulation Perspective

The **Simulation** perspective opens after you launch a simulation; you are prompted to switch to the **Simulation** perspective. Switching to the **Simulation** perspective means that you can now connect to your running nodes in the **Simulations** view. By default, it incorporates the **Topology Editor**, **Projects** view, **Simulations** view, **Console** view, and **Terminal** view.

From the **Simulation** perspective, right-click the **Simulation** icon. This, displays the following menu options:

Table 5: Simulation Perspective Context Menu Options

Operation	Description
Save As	Saves the current perspective.
Reset	Resets the current perspective to its original configuration (when the workbench opened initially). **Note** Any open editors, for example, **Topology Editor**, are not closed when a perspective gets reset to its original configuration. They are stacked as tabs in the default editor area for that perspective.
Close	Closes the current perspective. (Reopen it using the **Open Perspective** tool.)
Show Text	Displays perspective icons only. (No text is displayed in the toolbar.)

Cisco Modeling Labs Client Views

Views provide alternative methods for presenting topology information and navigating within your workbench. You can drag the view windows and position them anywhere within the workbench. Some views have their own toolbars, and some of the tools on these toolbars are specific to the views being presented.

The most commonly used views within the Cisco Modeling Labs client are listed in the following table:

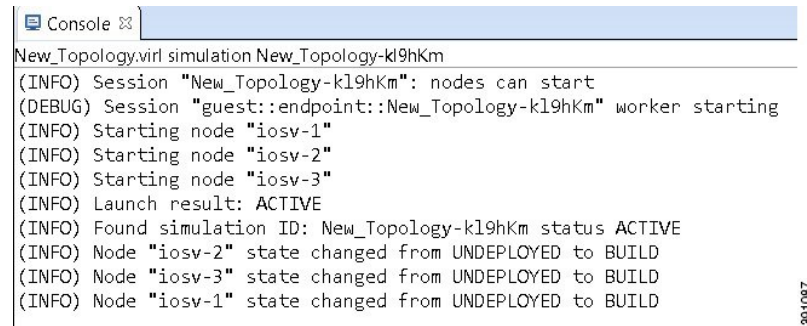*Table 6: Commonly Used Views in the Cisco Modeling Labs Client*

View Name	Perspective Where Used	Description
Console view	Design and Simulation	**Design**—Displays messages from AutoNetkit when it is used to generate router configurations. **Simulation**—Displays message streams from the Cisco Modeling Labs server after a simulation is launched. For more information, see Console View, on page 17
Graph Overview	Design	Provides you with the ability to view a scaled-down version of your entire topology. For more information, see Graph Overview, on page 18

View Name	Perspective Where Used	Description
History view	Design	Provides a history of changes made to a file and enables you to select a previous file version from the list. For more information, see History View, on page 18
Outline view	Design	Shows a list of the nodes in the topology and their interface connections. For more information, see Outline View, on page 20
Palette view	Design	Allows you to add devices and interface connections to your topology. For more information, see Palette View, on page 21
Problems view	Design	Displays errors, warnings, and other information that was detected within the **Topology Editor**. For more information, see Problems View, on page 23
Projects view	Design and Simulation	Provides a hierarchical view of topology projects, folders, and topologies in the workbench. For more information, see Projects View, on page 28
Properties view	Design	Displays names and properties of nodes and interfaces. For more information, see Properties View, on page 29
Search view	Design	Displays the results of a search, which can be based on text strings, regular expressions, and patterns, whole words and case-sensitive characters. For more information, see Search View, on page 38
Simulations view	Simulation	Displays information on all running simulations. For more information, see Simulations View, on page 40
Terminal view	Simulation	Displays console information when you connect to a node using Telnet. For more information, see Terminal View, on page 45

Console View

The **Console** view displays message streams from the Cisco Modeling Labs server after a simulation is launched. It also displays messages from AutoNetkit when it is used to generate router configurations.

Figure 5: Console View

```
Console ✕
New_Topology.virl simulation New_Topology-kl9hKm
(INFO) Session "New_Topology-kl9hKm": nodes can start
(DEBUG) Session "guest::endpoint::New_Topology-kl9hKm" worker starting
(INFO) Starting node "iosv-1"
(INFO) Starting node "iosv-2"
(INFO) Starting node "iosv-3"
(INFO) Launch result: ACTIVE
(INFO) Found simulation ID: New_Topology-kl9hKm status ACTIVE
(INFO) Node "iosv-2" state changed from UNDEPLOYED to BUILD
(INFO) Node "iosv-3" state changed from UNDEPLOYED to BUILD
(INFO) Node "iosv-1" state changed from UNDEPLOYED to BUILD
```

The **Console** view toolbar contains the following tools:

Figure 6: Console View Toolbar

Table 7: Available Tools

Icon	Function	Description
	Clear Console	Removes all the information from the **Console** view.
	Scroll Lock	Switches scrolling on and off.
	Pin Console	Pins the **Console** view to the workbench so that subsequent message streams are shown in another **Console** view. The pinned view remains unchanged.
	Display Selected Console	Displays the **Console** view for the selected simulation.
	Open New Console	Opens a new **Console** view.

| Icon	Function	Description
	Minimize	Reduces the size of the **Console** view.
	Maximize	Increases the size of the **Console** view.

Note When several simulations are running, use the toolbar button **Display Selected Console** to toggle between the **Console** views for the different simulations.

Graph Overview

Graph Overview enables you to view a scaled-down version of your entire topology. A blue rectangle (representing an overlay) is used to indicate a portion of the topology that is currently being displayed in the **Topology Editor**. Using this overlay, you can easily see where the displayed portion sits in relation to the entire topology.

The **Graph Overview** also allows you to navigate around a large topology when it is either too large to fit into the canvas, or is zoomed in and not fully displayed on the canvas.

Figure 7: Graph Overview
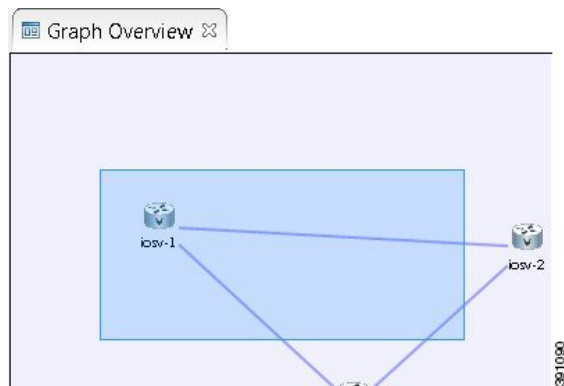
From the **Graph Overview**, click and drag the overlay to pan around your topology. As you drag the overlay, the corresponding content is reflected in the **Topology Editor**.

History View

When you create or modify a file, a history of record is maintained and a copy of the modified file is stored locally. This allows you to replace the current file with a previous version, or restore a file that has been

deleted. You can also use the **History** view to compare the changes that were made to the local files. Each file's history, which is stored locally, is uniquely represented by the date and time at which the file was saved.

Note Only changes made to topology files (.virl) are retained locally, changes made to projects and folders are not.

Figure 8: History View
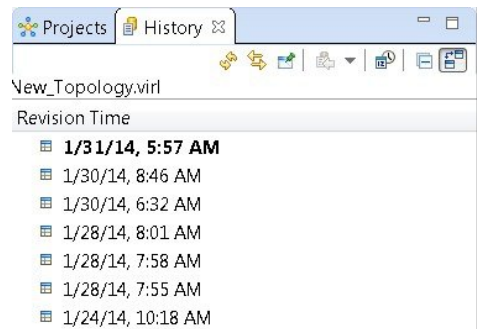
To view the change history of a file, in the **Projects** view, right-click the applicable file and choose **Team** > **Show Local History**.

The **History** view displays a list of revision times, with the most recent one highlighted at the top of the list.

Note If you have a .virl file opened in the workbench, click the **History** tab to view the list of changes made to the file.

The **History** view toolbar contains the following tools:

Figure 9: History View Toolbar

Table 8: Available Tools

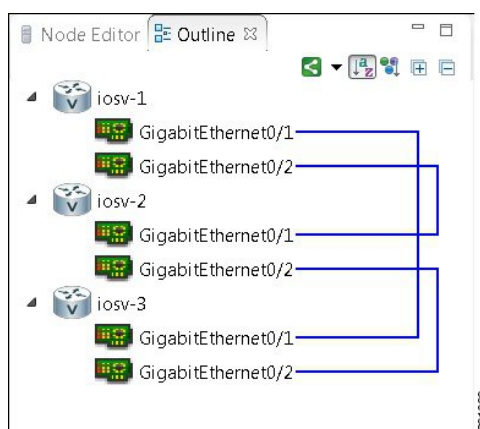| Icon	Function	Description
	Refresh	Refreshes the contents of the view, retrieving the latest history information for a file from the system.
	Link with Editor and Selection	Toggles when the **History** view selection is linked to the active editor. When this option is selected, changing the active editor automatically updates the **History** view selection to the project, folder, and file being edited.

Icon	Function	Description
	Pin this History View	Pins the view to the workbench and captures a snapshot of the file history information. New requests for file history are opened in a new instance of the **History** view.
	Group Revisions by Date	Sorts all history items by date. Options are: • **Today** • **Yesterday** • **This Month** • **Previous**
	Collapse All	Collapses all the history items listed in the hierarchical view.
	Compare Mode	Opens the compare editor for file comparison.

Outline View

The **Outline** view shows a list of the nodes in the topology and their interface connections. Node connectivity is shown through their interface connections.

Figure 10: Outline View

The **Outline** view toolbar contains the following tools:

Figure 11: Outline View Toolbar

Table 9: Available Tools

Icon	Function	Description
	Change Display Mode of Connection	Allows you to choose various options when displaying connections: • **Hide Connections**—No connections are shown. **Show Selected Connections**—Displays only the connections that are selected in the **Topology Editor**. • **Show Intra-Topology Connections**—Displays only the connections between the sites in the topology. • **Show Inter-Topology Connections**—Displays only the connections that are within a site. • **Show All Connections**—Displays all the connections for all the nodes in the topology.
	Sort Alphabetically	Sorts and lists the node names in alphabetical order.
	Sort By Node Type	Sorts and lists the node types in alphabetical order.
	Expand All	Expands nodes, interfaces, and connections in the hierarchical view.
	Collapse All	Collapses all of the items in the hierarchical view.

Palette View

The **Palette** view allows you to add devices and interface connections to your topology. Using the **Palette** view, you can:

• Add nodes, multipoint connections, L3 external Secure Network Address Translation (SNAT) connections, and L2 external (Flat) connections to your topologies.

• Select nodes and connections for repositioning on the canvas.

• Create connections between node interfaces.

• Group nodes into sites.

Figure 12: Palette View
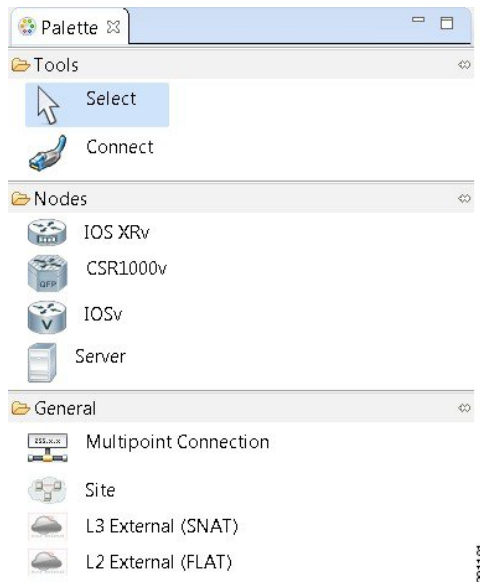
The **Palette** view is divided into the following categories:

• **Tools**—Contains the **Select** and **Connect** tools. The **Select** tool allows you to select nodes, multipoint connections, L3 external (SNAT) connections, and L2 external (Flat) connections on the **Topology Editor** canvas. The **Connect** tool creates connections between node interfaces.

> **Note** The **File** > **Preferences** > **Topology Editor** setting affects how the nodes and connections are placed on the canvas. If you check the **Revert back to the palette's default tool**, check box, you must click a node, connection, or other object each time you place an object on the canvas. If the **Revert back to the palette's default tool**, check box is not checked, each time you click the canvas, an object is placed until you click the **Select** tool (the default palette tool).

• **Nodes**—Contains the node types available for use in topologies. Currently, Cisco Modeling Labs 1.0 includes the Cisco IOSv virtual image.
An Ubuntu 14.04 server has been tested with Cisco Modeling Labs 1.0. The Ubuntu server must be downloaded separately. To access the server, go to:

http://cloud-images.ubuntu.com/trusty/current.

From this page, download the clouding-amd64 RAW (img) image **trusty-server-cloudimg-amd64-disk1.img**.

Note   Additional node subtypes can be installed separately. See the *Release Notes for Cisco Modeling Labs 1.0* for the most up-to-date list of supported virtual images.

- **General**—Contains the different types of connection functions that are supported for nodes. Options are:

  ◦ Multipoint connections

  ◦ Layer 3 (L3) external connections

  ◦ Layer 2 (L2) external connections

  You can also create sites.

The **Palette** view toolbar contains the following tools:

Figure 13: Palette View Toolbar

Table 10: Available Tools

Icon	Tool	Description
	Minimize	Reduces size of **Palette** view.
	Maximize	Increases size of **Palette** view.

Problems View

The Topology Editor automatically detects errors, warnings, or other information that are displayed on the topology elements in the Topology Editor and Node Editor. These markers can be viewed in the **Problems** view.

Note By default, the **Problems** view displays all the errors and warnings for all the topologies in the **Projects** view, not just the currently open topology. From the **Problems** view toolbar, choose **View Menu** > **Show** > **Errors/Warnings on Project** to filter only those errors and warnings that are applicable to the current project. Alternatively, you can also use the **Configure Contents** dialog box, which is accessible from the **View Menu** option, to filter warnings and errors associated with a particular topology or topology project. See **Problems View Toolbar** for more information.
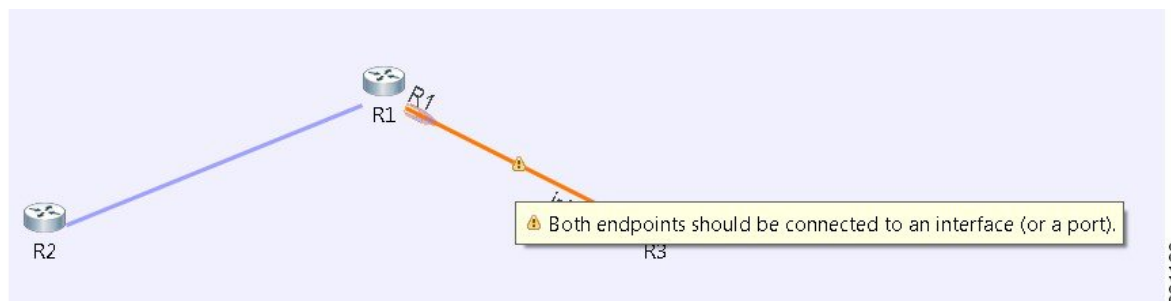
Figure 14: Problems View

The **Problems** view groups errors and warnings by severity, with the most critical issues listed first.

Double-click the problem marker in the **Problems** view, which opens the appropriate editor in the Cisco Modeling Labs client. If the problem relates to an XML file, the XML file opens in a text editor. The problem is highlighted, allowing you to quickly identify the issue and correct it.

Figure 15: Problem Example

The **Problems** view toolbar contains the following tools:

Figure 16: Problems View Toolbar

Table 11: Available Tools

Icon	Function	Description
	View Menu	The **Problems** view tool has the following options: • **Show**—Displays errors and warnings. • **Group By**—Groups problems under the headings **Type**, **Severity**, and **None**. The default is **Severity**. • **Sort By**—Sorts problems under the following headings: ◦ **Description**—Details the problem encountered. ◦ **Resource**—Displays the name of the .virl file where the problem has occurred. ◦ **Path**—Displays the applicable project folder. ◦ **Location**—Displays the location in the .virl file where the problem has occurred. ◦ **Type**—Displays the type of problem, for example, XML problem. • **New Problem View**—Opens a new **Problems** view on the workbench. • **Configure Contents**—Opens the **Configure Contents** dialog box where you can add multiple filters to the **Problems** view and enable or disable them, as required. Filters can either be additive or exclusive. The **All Errors/Warnings on Selection** filter is provided by default. • **Configure Columns**—Opens the **Configure Columns** dialog box where you can choose to hide or show specific information about the problem encountered, as shown in the **Sort By** option. Options are: ◦ **Creation Time**—Displays the time when the problem occurred. ◦ **Description, ID**—Displays the system-generated ID for the problem, location, path, resource, and type.
	Minimize	Reduces the size of the **Problems** view.
	Maximize	Increases the size of the **Problems** view.

The Quick Fix Option

Problems displayed in the **Problems** view are provided with a **Quick Fix** option, if available. A Quick Fix is indicated by a light-bulb icon that is visible on the marker. When this option is selected, you are presented with one or more possible fixes.

We recommend that you use Quick Fix to resolve the errors discovered, unless the errors have been deliberately created for testing purposes.
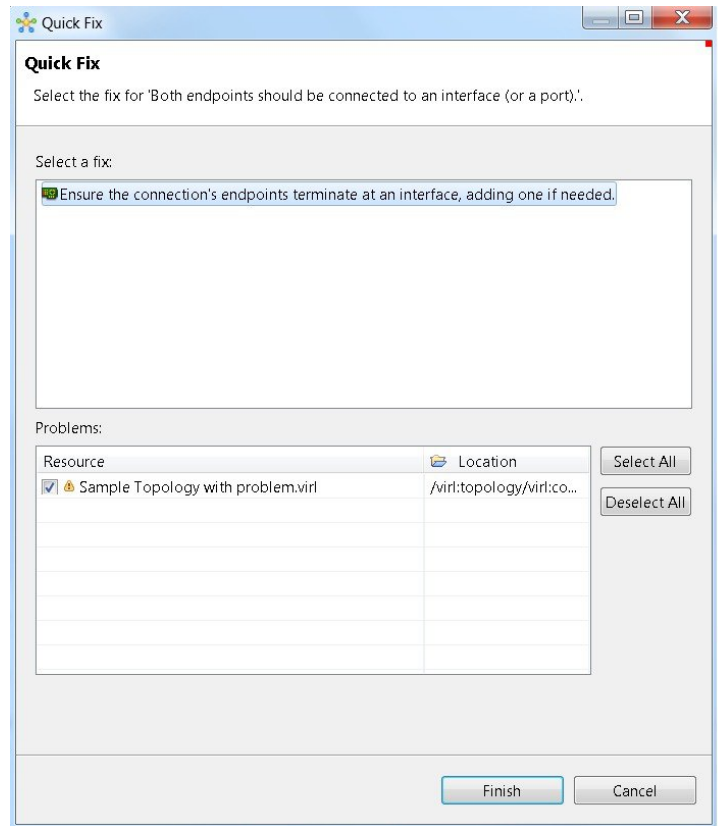
Figure 17: Quick Fix
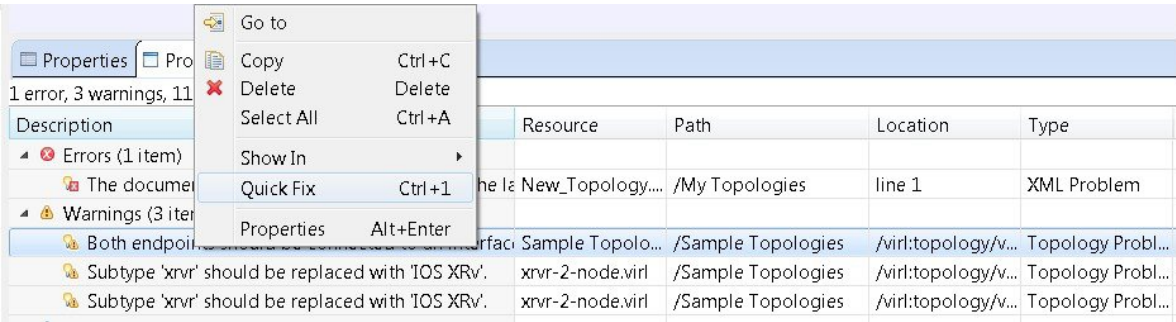
To fix a problem:

1 Right-click the line containing the problem, and select **Quick Fix**.

The **Quick Fix** dialog box displays a list of possible solutions.

Figure 18: Quick Fix Dialog Box

2   Select a fix from the list, and then check the check box of any of the Resources listed in the **Problems** area. You can click **Select All** to apply the Quick Fix to all the Resources listed. Alternatively, you can click **Deselect All** to clear all selections.

3   Click **Finish**.

Note   Once a problem has been fixed using the Quick Fix option, the action cannot be undone.

Projects View

The **Projects** view provides an hierarchical view of topology projects, folders, and topologies in the workbench. From here, you can open topologies for editing, or select resources for operations, such as exporting.

Figure 19: Projects View

Right-click any topology in the **Projects** view to open a context menu using which you can copy, move, and create new topology files, view comparison files, and so on.

The **Projects** view toolbar contains the following tools:

Figure 20: Projects View Toolbar

Table 12: Available Tools

Icon	Function	Description
	Collapse All	Collapses the hierarchy of all the resources in the **Projects** view.
	Link with Editor	Links the **Projects** view with an active editor. A change to an active editor automatically updates the **Projects** view and allows you to toggle between the two views.
	View	Provides options for customizing the content displayed in the **Projects** view. Options are: • **Sort by filters** • **Sort by content**
	Minimize	Reduces the size of the **Projects** view.

Icon	Function	Description
	Maximize	Increases the size of the **Projects** view.

The **Projects** view displays several icons on the toolbar:
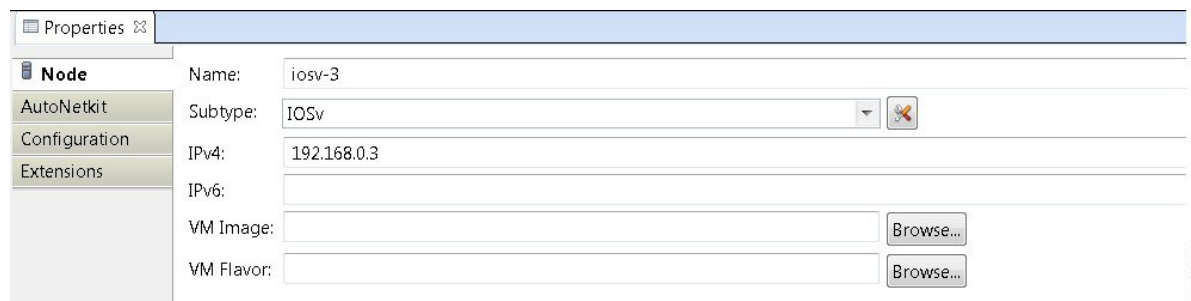
Table 13: Project View Icons

Icon	Name	Description
	Topology Project	Indicates an open topology project.
	Folder	Indicates an open folder. Folders are created within a Topology Project so that topology files can be organized into separate areas for greater accessibility.
	Topology	Indicates a topology file.

Properties View

The **Properties** view displays the names and properties of nodes and interfaces. If no specific node or interface is selected, the Properties settings apply globally to all the nodes and interfaces within a topology. If a specific node or interface is selected, the Properties settings apply to only that node or interface.

Figure 21: Properties View

The **Properties** view toolbar contains the following tools:

Figure 22: Properties View Toolbar

Table 14: Available Tools

Icon	Function	Description
	Show Categories	Shows the available property categories.
	Show Advanced Properties	Shows all advanced properties.
	Restore Default Value	Restores the default value for the property.
	Pin to Selection	Pins this property view to the current selection.
	View Menu	Displays menu items that allow you to: • Open a new property view. • Pin the property view to the current selection.
	Minimize	Reduces the size of the **Properties view**.
	Maximize	Increases the size of the **Properties view**.

The properties in the **Properties** view are discussed in the following sections:

- Node Properties, on page 31
- Topology Properties, on page 33

Node Properties

When you select a node on the canvas, the properties for that node are displayed in the **Properties** view.

Figure 23: Node Properties

Under the **Node** tab, you can perform the following tasks:

Table 15: Node Properties

Property	Description
Name	Specify a name for the node. **Note** Use only alphanumeric characters for node names. Node names must be unique across the entire topology. Duplicate node names cause the build to fail when the configuration is autogenerated. If a duplicate node name is defined, a marker is shown in the **Problems** view. Unicode is not supported for node names. The use of a period in a node name may cause the node name to wrap when viewing the hierarchy from the Cisco Modeling Labs server. When you update a node name in the **Node Properties** view, the node name is not updated in the **Node Editor**.
Subtype	Specify a subtype from the list. Cisco Modeling Labs 1.0 includes a Cisco IOSv virtual image. Additional Cisco virtual images are available for use. However, they must be installed separately. For a list of supported subtypes, see the *Release Notes for Cisco Modeling Labs 1.0*. An Ubuntu 14.04 server has been tested with Cisco Modeling Labs 1.0. The Ubuntu server must be downloaded separately. To access the server, go to: http://cloud-images.ubuntu.com/trusty/current. From this page, download the clouding-amd64 RAW (img) image **trusty-server-cloudimg-amd64-disk1.img**. Use the **Manage Subtypes** tool to add or remove node subtypes from the list. **Note** When changing a node subtype after it is initially configured using AutoNetkit, for example, changing it from Cisco IOSv to Cisco IOS XRv, you need to re-validate the AutoNetkit properties for the new node subtype.
IPv4	Specify an IPv4 loopback address. The loopback address is added to the router as interface loopback0. Enter a valid IP address in the correct format.
IPv6	Specify an IPv6 loopback address. The loopback address is added to the router as interface loopback0. Enter a valid IP address in the correct format.

Property	Description
VM Image	Specify a virtual machine (VM) image other than the default. Click **Browse** to choose a valid VM image from the **Select VM Image** dialog box.
VM Flavor	Specify a VM flavor other than the default. Click **Browse** to choose a valid VM flavor from the **Select VM Flavor** dialog box.

Under the **AutoNetkit** tab, when you check the **Auto-generate the configuration based on these attributes** check box, the AutoNetkit generates the configuration for your topology when you click the **Update Router Configurations** icon on the toolbar.

Note   Any pre-existing configuration for a node is overwritten when you choose **Update Router Configurations** from the toolbar. Uncheck this **Auto-generate** check box if you do not want router configuration for a node updated by AutoNetkit.

Using the properties listed, you can perform the following tasks:

Table 16: AutoNetkit Properties

Property	Fields	Description
General	**ASN**	Specify the autonomous system number, which is used to infer IGP and BGP and should be any valid integer.
IGP	**OSPF**	Configure an OSPF area. The default value is **0**.
iBGP	**iBGP Role**	Specify an iBGP role from the list and use it to create an iBGP topology. Options are: • **Not specified** • **Disabled** • **Peer** • **Route Reflector (RR)** • **Hierarchical Route Reflector (HRR)** • **Route Reflector Client (RRC)** The default value is **Peer**.
	RR Cluster	Specify the RRC as a name or number. Should be an alphanumeric string.
	HRR Cluster	Specify the HRR Cluster. Should be an alphanumeric string.
MPLS	**VRF Name**	Specify an MPLS VPN name for VRF Lite.

Property	Fields	Description
	Enable MPLS TE	Configure MPLS Traffic Engineering. Options are: • **Unspecified** • **True** • **False** The default value is **False**.
Server	**Server Username**	Specify a username for the server.
	SSH Key	Specify a secure shell key for the server. **Note** An SSH key is required only if you want to connect to the VM server within the topology using an external application. Using the VNC option does not require an SSH key to be specified.

Information displayed under the **Configuration** tab, depends on whether or not the **Auto-generate the configuration based on these attributes** check box under the AutoNetkit tab is checked.

- When checked, AutoNetkit generates the configuration and displays it under the node's **Configuration** tab.

- When unchecked, no configuration information is created for the node. You must configure the node manually, or cut and paste the existing configuration information into this area.

Under the **Extensions** tab, all the extensions used to generate the configuration are listed along with **Key**, **Value**, and **Type** attributes.

Topology Properties

When you select an area on the canvas other than a node, the properties for that topology are displayed in the **Properties** view.

Figure 24: Topology Properties
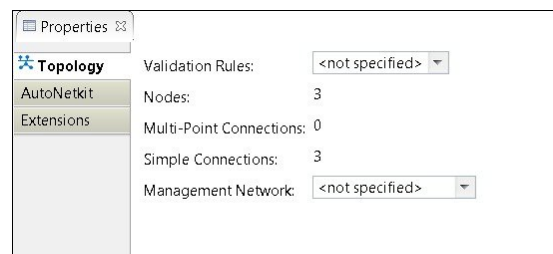
Under the **Topology** tab, you can perform the following tasks:

Table 17: Topology Properties

Property	Description
Validation Rules	Specify the rule type with which the topology is validated. Options are: • **Not specified** • **VXR** • **VIRL** For Cisco Modeling Labs 1.0, we recommend you accept the default value, **Not specified**.
Management Networks	Specify the type of Out of Band (OOB) external network access. Options are: • **Not specified**—No OOB external network access • **Shared Flat Network**—Enables OOB external network access to all the devices in the topology • **Private User Network**—Enables OOB external network access using Network Address Translation (NAT)

The **Nodes**, **Multipoint Connections**, and **Simple Connections** properties are for information purposes only and are useful when monitoring the size of the topology to your current user quotas or licensing limits or both.

Note Nodes in this instance includes node subtypes, Flat, and SNAT port groups.

Under the **AutoNetkit** tab, you can perform the following tasks:

Table 18: AutoNetkit Properties

Property	Fields	Description
General	**Enable CDP**	Configure the Cisco Discovery Protocol (CDP). Options are: • **True** • **False** The default value is **False**.
	Enable onePK	Configure the Cisco One Platform Kit (onePK). Options are: • **True** • **False** The default value is **False**.

Property	Fields	Description
Addressing	**IP Address Family**	Configure a routing session to use IPv4 address prefixes, IPv6 address prefixes, dual stack (includes both native IPv4 and IPv6), or none. Options are: • **Not specified** • **None** • **v4** • **v6** • **dual_stack**
	IPv4 Infrastructure Subnet	Specify the address to use for IPv4 infrastructure address allocations. This is the address assigned to the interface created on the router. The default value is 10.0.0.0. Enter a valid IP address in the correct format.
	IPv4 Infrastructure Prefix	Specify the prefix to use for IPv4 infrastructure address allocations. The default value is 8. Enter a valid IP address in the correct format.
	IPv4 Loopback Subnet	Specify the address to use for IPv4 loopback address allocations. The default value is 192.168.0.0. Enter a valid IP address in the correct format.
	IPv4 Loopback Pool	Specify the prefix size to use for IPv4 loopback address allocations. The default value is 22. Enter a valid IP address in the correct format.
	IPv4 VRF Subnet	Specify the address to use for IPv4 VRF address allocations, when specifying the address range for MPLS VRF. The default value is 172.16.0.0. Enter a valid IP address in the correct format.
	IPv4 VRF Prefix	Specify the prefix to use for IPv4 VRF address allocations. The default value is 24.
Routing	**Enable Routing Protocols**	Configure routing protocols (BGP and IGP). Options are: • **Not specified** • **True** • **False** The default value is **True**. If you specify **False**, there will be no router configuration for any of the routing protocols.
	IGP	Configure the Interior Gateway Protocol (IGP). Options are: • **OSPF** • **ISIS** • **EIGRP**

Property	Fields	Description
MPLS	Enable MPLS OAM	Configure MPLS OAM for all routes on the topology. Options are: • **Not specified** • **True** • **False** The default value is **False**.

The following table shows the default IP Address values used by Cisco Modeling Labs. You can update these values as required.

Table 19: IP Address Default Values

Viewed from	Option	Default Value	Optional Value(s)
Topology > AutoNetkit > Addressing	IP Address Family	v4	None, v6, dual_stack
	IPv4 Infrastructure Subnet	10.0.0.0	
	IPv4 Infrastructure Prefix	8	
	IPv4 Loopback Subnet	192.168.0.0	
	IPv4 Loopback Pool Prefix	22	
	IPv4 VRF Subnet	172.16.0.0	
	IPv4 VRF Prefix	24	

In Cisco Modeling Labs 1.0, AutoNetkit IPv6 handling code does not support the following:

• The ability to manually specify IPs.

• The ability for users to specify the IPv6 range in the Cisco Modeling Labs client.

• The ability to divide out the allocation logic to permit both infrastructure and loopback to be automatically or manually created for either combination.

The following table shows the default routing protocols used by Cisco Modeling Labs. You can update these values as required.

Table 20: Routing Protocols Default Values

Viewed from	Option	Default Value	Optional Value(s)
Topology > AutoNetkit > General	Enable CDP	false	true
	Enable OnePK	false	true

Viewed from	Option	Default Value	Optional Value(s)
Topology > AutoNetkit > Routing	**Enable Routing Protocols**	true	false
	IGP	ospf	isis, eigrp
Topology > AutoNetkit > MPLS	**Enable MPLS OAM**	false	true
Node > AutoNetkit > General	**ASN**	1	None or any valid integer
Node > AutoNetkit > IGP	**OSPF Area**	0	None or valid OSPF area number
Node > AutoNetkit > iBGP	**iBGP Role**	Peer	Disabled, RRC, HRR, RR
	RR Cluster	(none)	None or alphanumeric string
	HRR Cluster	(none)	None or alphanumeric string
Node > AutoNetkit > MPLS	**VRF Name**	(none)	None or alphanumeric string
	Enable MPLS TE	false	true
Node > AutoNetkit > Server	**Server Username**	(none)	Valid username
	Server SSH Key	(none)	(none)

Note   Only one IGP protocol is supported per simulation. Even though you can create multiple BGP ASs, the choice of IGP applies to all router devices in the simulation.

Under the **Extensions** tab, all the extensions used to generate the configuration are listed, along with the **Key**, **Value**, and **Type** attributes.

Interface Properties

In the **Node Editor**, selecting an interface on the canvas displays properties for the interface in the **Properties view**.

Under the **Interface** tab, you can perform the following tasks:

Table 21: Interface Properties

Property	Description
Name	Specify a name for the interface.
IPv4	Specify an IPv4 interface address and an IPv4 subnet prefix length.

Property	Description
IPv6	Specify an IPv6 interface address and an IPv6 subnet prefix length.

Note   To delete an interface from the **Node Editor** canvas, select the interface, right-click and choose **Delete** from the context menu.

Under the **Extensions** tab, all the extensions used to generate the configuration are listed, along with the **Key**, **Value**, and **Type** attributes.

Connection Properties

When you select a connection on the canvas, the properties of that connection are displayed in the **Properties** view.

From **Connection** properties, you can associate a line style design with a connection between nodes. Line styles are visual aids that help you identify the connections used in your topology design.

Search View

To search for text string and files, click **Search** on the toolbar. The **Search** view displays the results of a search. A file search can be based on text strings, regular expressions, and patterns, in addition to whole words and case-sensitive characters. The scope of a file search can encompass a workspace, selected resources, or projects.

Note   Text searches are only performed on expressions contained in files with the extension .virl.

The **Search** view also displays the results of a Git search that can be based on the same criteria as noted in a File search. However, the scope of a Git search can encompass a particular message, resource, or identification number in the code.

Search criteria can be based on either a file or Git. The scope of a search can be general to all the topologies that are defined on Cisco Modeling Labs client, or specific to a particular project or file.

Figure 25: Search View
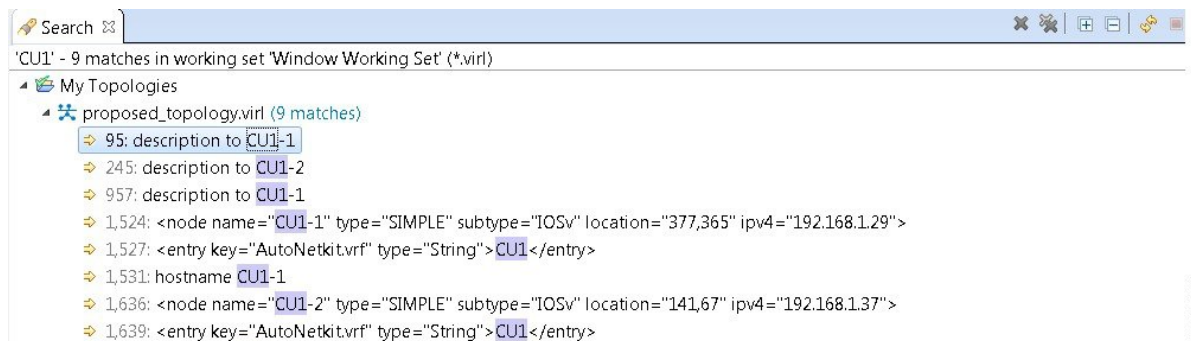
To use the search functionality, perform the following tasks:

1 From the Cisco Modeling Labs client toolbar, click **Search** > **File**.
The **Search** dialog box appears.

2 In the **Containing text** field, enter the text string to search for. The Containing text field displays a list of recently performed searches to select from. Leave this field empty if you want to search for files only. Check or uncheck the **Case sensitive** check box depending on whether or not a case-sensitive search is to be performed. You can also check the **Regular expression** check box to enable more powerful searching capabilities. Check the **Whole word** check box if you want to search for whole words that are identical to the text string. Specify the types of files to include in the search in the **File name patterns** field.

3 Click **Choose** to open the **Select Types** dialog box. This dialog box provides a quick way to select from a list of valid extensions.

4 In the **Scope** area, specify the files and folders to include in the search. Valid options are:

  • The entire workspace

  • The currently selected resources in the workspace

  • A named working set

  • A customized group of files and folders. Use the **Customize** option to define the type of available searches from the **Search Page Selection** dialog box.

5 Click **Search** to begin your search.
The **Search** view appears with the results of the search listed. You can click the **Cancel** tool in the **Search** view to cancel your search while it is still in progress.

The **Search** view toolbar contains the following tools:

Figure 26: Search View Toolbar

Table 22: Available Tools

Icon	Function	Description
	Remove Selected Matches	Deletes all the highlighted matches from the search results.
	Remove All Matches	Deletes all the matches from the search results.
	Expand All	Expands each item in the **Search** view.

Icon	Function	Description
	Collapse All	Collapses each item in the **Search** view.
	Run the Current Search Again	Reruns the current search to retrieve previous search results or to reflect recent changes.
	Cancel Current Search	Cancels the search currently running.
	Show Previous Searches	Browses previously conducted searches to repeat a previous search, by selecting a previous search from the drop-down menu. You can also clear the search history.
	Pin the Search View	Pins the **Search** view so that subsequent search results are displayed in a separate **Search** view with the pinned view unchanged, in order to compare results.
	View Menu	Displays the search results as a tree or a list, filters the results using the **Filters** option, and sets the overall preferences for search using the **Preferences** option.
	Minimize	Reduces the size of the **Search** view.
	Maximize	Increases the size of the **Search** view.

Simulations View

The **Simulations** view displays information about all the running simulations, including:

- Name of the user running the simulation
- Name of the topology
- Number of nodes in the running simulation

• Current state of each node

Figure 27: Simulations View
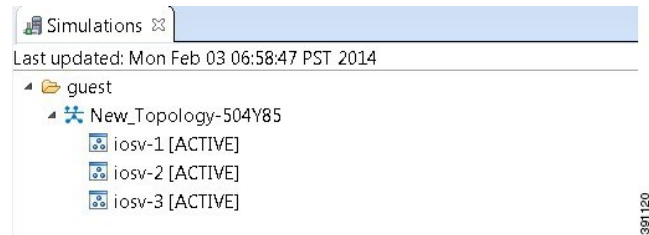
Possible simulation states are:

Table 23: Simulation States

State	Description
ACTIVE	Indicates that the launch worker process has successfully made all requests to OpenStack to deploy all simulation nodes.
DONE	Indicates that a simulation has been successfully stopped and no longer exists in the STD database.
FAIL	Indicates that the launch worker process failed.
INIT	Indicates that the simulation has just been created and is being launched.
LAUNCHING	This is a two-phase state: i) Necessary DB entries are created in the middleware and ii) Required networks are deployed. Failure in this phase results in the removal of the simulation; success results in the start of a launch worker process.
STOP	Indicates that a stop simulation request has been received.

Possible node states are:

Table 24: Node States

State	Description
ACTIVE	Indicates that the virtual machine process is successful. It may take a few minutes for the node to boot up and configure.
BUILD	Indicates that the virtual machine is starting, but the router image has not yet loaded.
DELETED	Indicates that the node has been requested for BUILD, but OpenStack has no information on the node at this time. Possibly, it was deleted. The node can be started when the simulation itself is ACTIVE.
ERROR	Indicates that the virtual machine process failed.

State	Description
IDLE	Indicates that the virtual machine is in an unused state.
PAUSED and **SUSPENDED**	These two states can occur when: • The host environment is rebooted. • The node itself shuts down. • The operating system of the node crashes . • The virtualization process running the node crashes. In such instances, the appropriate action is to stop the nodes and then start the nodes again. **Note** All configuration changes to the nodes are lost .
SHUTOFF	Indicates that the virtual machine is deployed but not actually running. This state can occur when: • The host environment is rebooted. • The node itself shuts down. • The operating system of the node crashes . • The virtualization process running the node crashes. In such instances, the appropriate action is to stop the nodes and then start the nodes again. **Note** All configuration changes to the nodes are lost.
UNDEPLOYED	Indicates that the node has never requested to be deployed in OpenStack (started) or the request failed. This only occurs when the launch worker process is active but has not reached that specific node yet, or if the simulation is in the FAIL state.

The **Simulations** view toolbar contains the following tools:

Figure 28: Simulations View Toolbar

Table 25: Available Tools

Icon	Function	Description
	Refresh the List	Refreshes the list of simulations displayed in the**Simulations** view.

Icon	Function	Description
	Minimize	Reduces the size of the **Simulations** view.
	Maximize	Increases the size of the **Simulations** view.

Topology Options

The following operations are available, when you right-click the topology name in the **Simulations view**:
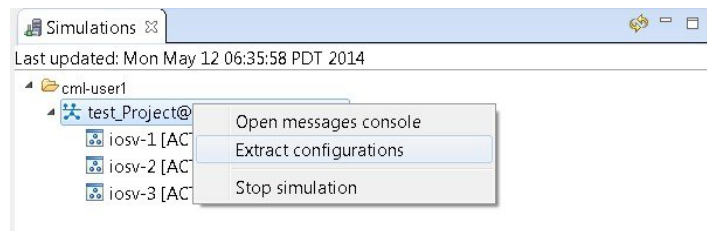
Figure 29: Topology Options

Table 26: Topology Options

Operation	Description
Open messages console	Opens the **Console** view, showing the message stream from Cisco Modeling Labs server for the selected topology. Message streams contain information on the topology launch, such as, the name of the launched topology, the date and time of the launch, and the current status of each node in the topology.
Extract configurations	Extracts router configurations for all the routers to a file saved locally. **Note** All active console connections are automatically disconnected by Cisco Modeling Labs. All external Telnet connections to the console ports must be closed manually. The **Stop simulation** operation cannot be selected prior to the **Extract configurations** operation.
Stop simulation	Stops the running simulation. See the chapter Simulate the Topology Overview for more information on stopping and starting simulations.

Node Options

The following operations are available, when you right-click the node name in the **Simulations** view:
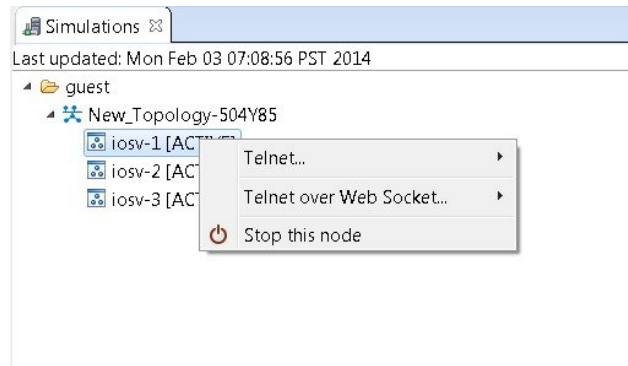
Figure 30: Node Options

Table 27: Node Options

Operation	Description
Telnet	Allows you to connect using Telnet to ports on a node.
Telnet over Web Socket	Allows you to connect using Telnet over a web socket to ports on a node. Web sockets provide full-duplex communications channels over a single connection.
Attach to its VNC	Displays a server Virtual Network Computing (VNC) login page. **Note** This operation applies to the server only.
Stop this node	Stops the selected node.
Start this node	Starts the selected node. See the section Start a Single Node for more information.

Terminal View

The **Terminal** view is displayed when you connect via Telnet to a node. Using the **Terminal** view, you can communicate with and control the operating system running on the node.

Figure 31: Terminal View

The **Terminal** view toolbar contains the following tools:

Figure 32: Terminal View Toolbar

Table 28: Available Tools

Icon	Function	Description
	Disconnect	Disconnects the terminal connection to the node.
	Scroll Lock	Sets scrolling on and off.
	Display Selected Connections	Allows you to select a connection from the list of active terminal connections.
	Remove Terminal	Closes the Terminal view.

Icon	Function	Description
Aa	**Set Terminal Font**	Allows you to set the font to be used in the terminal, from the **Colors and Fonts** dialog box. **Note** You can also access the **Colors and Fonts** dialog box by choosing **File** > **Preferences** > **General** > **Appearance** > **Colors and Fonts**.

Setting Preferences for the Cisco Modeling Labs Client

For the Cisco Modeling Labs client to operate, you must first identify certain setting preferences. These preferences are available from the menu bar under **File** > **Preferences**:

These are discussed in the following sections.

Node Subtypes Setting

A node subtype defines a virtual machine running a particular operating system. For example, Cisco IOSv is installed with Cisco Modeling Labs, while others such as Cisco IOS XRv and Cisco CSR1000v can be installed separately. This setting allows you to add and remove node subtypes for use with the Cisco Modeling Labs client.

Refer to the chapter *User Workspace Management* in the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide* for information on installing virtual machine images on the Cisco Modeling Labs server.

Figure 33: Node Subtypes Setting
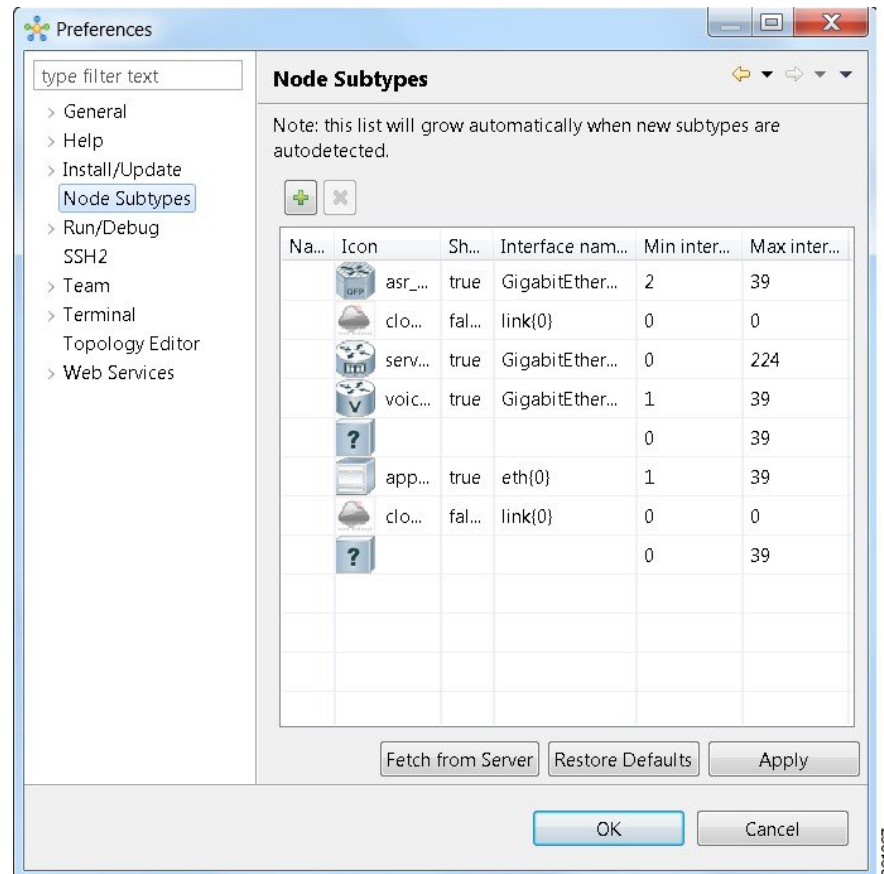
The available operations for this setting are:

Table 29: Node Subtypes Setting Operations

Icon	Operation	Description
	Add new Subtype	Adds a new subtype to the list. To ensure that the new subtype is visible in the **Palette** view, set the value in the **Show in Palette** column to **true**. See Adding a New Node Subtype to the List, on page 48.
	Remove Subtype	Removes a subtype from the list.

Icon	Operation	Description
	Fetch from Server	Updates the local subtypes based on the currently configured Cisco Modeling Labs server. All the subtypes supported on Cisco Modeling Labs server are available with this operation. See Fetching Node Subtypes from the Cisco Modeling Labs Server, on page 49.
	Restore Defaults	Reverts to the original list of subtypes.
	Apply	Applies changes.

Adding a New Node Subtype to the List

To add a new node subtype to the node subtype list, perform the following tasks:

Step 1   Click **File** > **Preferences** > **Node Subtypes**.

Step 2   Click the **Add new subtype** button.

a) In the **Name** column, enter a name for the new node subtype, for example, Server_Block1.

b) In the **Icon** column, select an icon type from the drop-down list, for example app_server.

c) In the **Show in Palette** column, click to select **True** to display the node subtype in the **Palette** view, else select **False**.

d) In the **Interface name format** column, enter a name for the interface format, for example eth{0}.

e) In the **Min interface** column, enter the minimum number of interfaces permissible for the node subtype, for example, 1.

f) In the **Max interface** column, enter the maximum number of interfaces permissible for the node subtype, for example, 25.

Figure 34: New Node Subtype Added
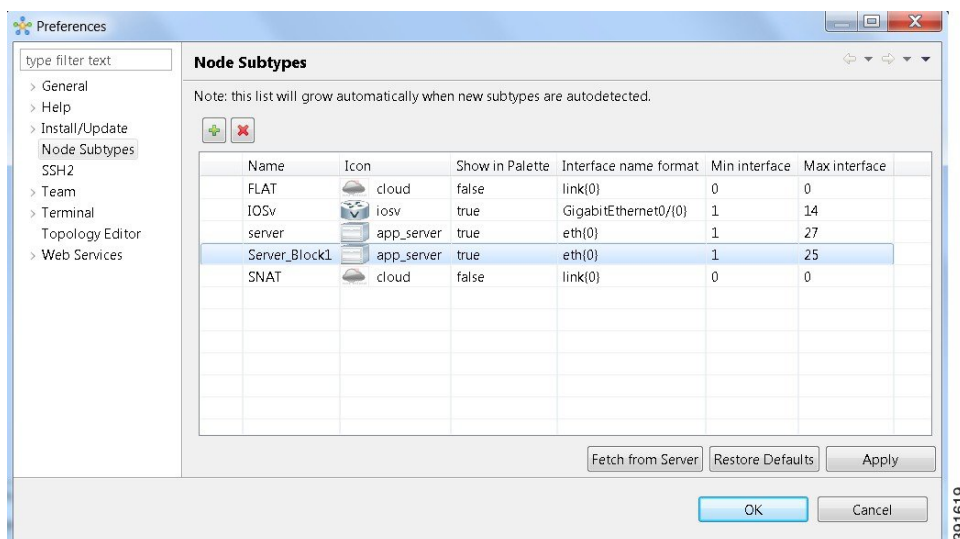
Step 3   Click **Apply** and **OK** to save the updates.

The newly added node subtype is available for use in the **Palette** view.

Figure 35: New Node Subtype Visible in the Palette View

Fetching Node Subtypes from the Cisco Modeling Labs Server

To fetch new node subtypes from the Cisco Modeling Labs server, perform the following tasks:

Step 1   Click **File** > **Preferences** > **Node Subtypes**.

Step 2   Click the **Fetch from Server** button.
The **Confirm** dialog box is displayed.

Step 3   Click **OK** to update the list of node subtypes.

Figure 36: Fetch Nodes Subtypes from Server
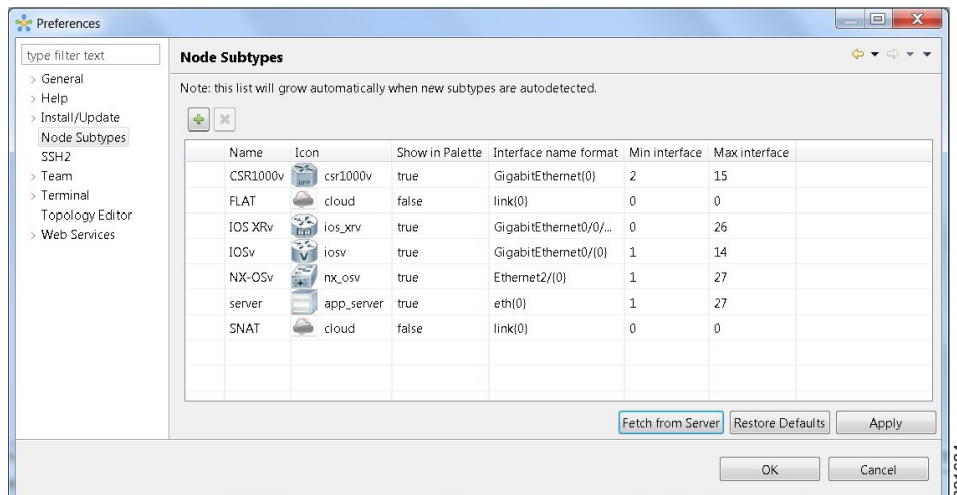
Step 4   Click **OK** to finish.

The updated list of node subtypes is available for use in the **Palette** view.

Figure 37: Updated List of Node Subtypes Visible in the Palette View

Contact your system administrator if a specific node subtype is missing from the list, as the system administrator is responsible for adding new node subtypes to the Cisco Modeling Labs server.

Terminal Setting

This setting allows you to launch an external terminal application, such as, SecureCRT or PuTTY, or use the internal Cisco Modeling Labs client **Terminal** view in a separate window.

Note

- If you are using the internal Cisco Modeling Labs **Terminal** view, the views are visible from both the **Design** or **Simulation** perspective. However, a detached view is only visible from the perspective in which it was detached.

- If you are using an external terminal application, you must specify both Telnet and SSH run commands. You must also ensure that the title format includes the percentage (%) character. Omission of either of these requirements will impede your ability to save your setting preferences.
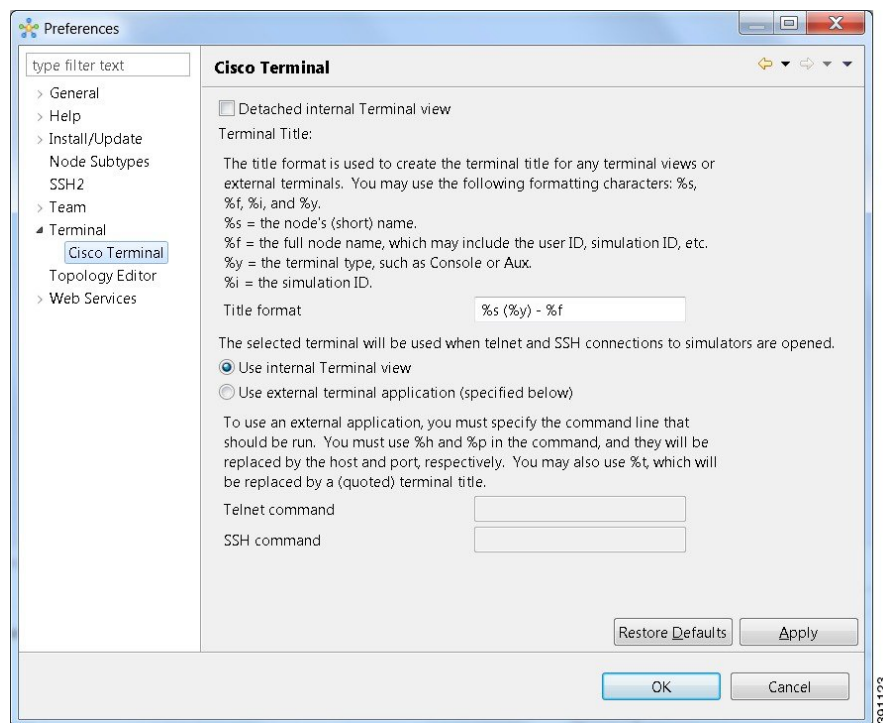
- When you specify to use an external terminal via **File** > **Preferences** > **Terminal** > **Cisco Terminal** and then Telnet over Web Socket to a virtual machine, the terminal opens internally, not externally as specified.

Figure 38: Terminal Setting

The available operations for this setting are:

Table 30: Terminal Setting Operations
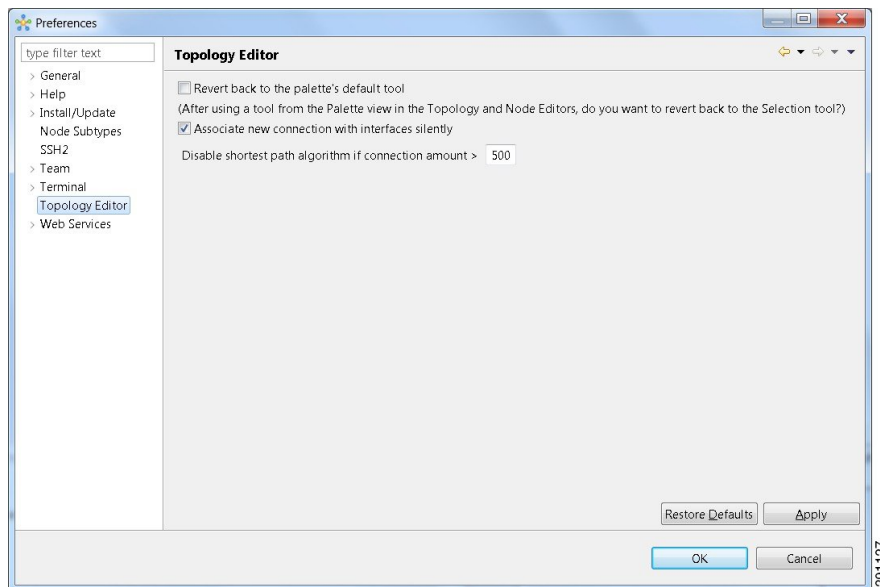
Operation	Description
Detached internal Terminal view	Launches the **Terminal** view in a separate window.
Title format	Allows you to add a **Terminal** view title using the required formatting characters.
Use internal Terminal view	Uses the Cisco Modeling Labs internal **Terminal** view.

Operation	Description
Use external terminal application	Uses an external terminal application, such as SSH or PuTTY.
Telnet command	Specifies the Telnet command to run if you are using an external terminal application.
SSH command	Specifies the SSH command to run if you are using an external terminal application.
Restore Defaults	Removes settings specified for an external terminal application and restores terminal settings to the Cisco Modeling Labs internal **Terminal** view.
Apply	Applies changes.

Topology Editor Setting

This setting allows you to customize the Topology Editor in the Cisco Modeling Labs client.

Figure 39: Topology Editor Setting

The available operations for this setting are:

Table 31: Topology Editor Setting Operations

Operation	Description
Revert back to the palette's default tool	Resets the definition of the **Palette** view tools, **Select** and **Connect** to their default configuration. This option is disabled by default.

Operation	Description
Associate new connection with interfaces silently	Enables the following: • When selected, interfaces are automatically assigned to connections between nodes. • When deselected, you are presented with a list of available interfaces.
Disable shortest path algorithm if connection amount > (greater than)	Enables the following: • When the number of connections in the topology exceeds the value set in this field, the internal algorithm used to connect nodes will not be the shortest path algorithm.
Restore Defaults	Restores settings to the initial default state.
Apply	Applies changes.

Web Services Setting

This setting allows you to configure the Cisco Modeling Labs client to communicate with the Cisco Modeling Labs server. When you first launch the Cisco Modeling Labs client, the Active profile is not specified and the Web Services that are listed display an Unauthorized message in red. This message relates to the Master

Credentials, which must be still be set before the Cisco Modeling Labs client can communicate with the Cisco Modeling Labs server.

Figure 40: Web Services Setting
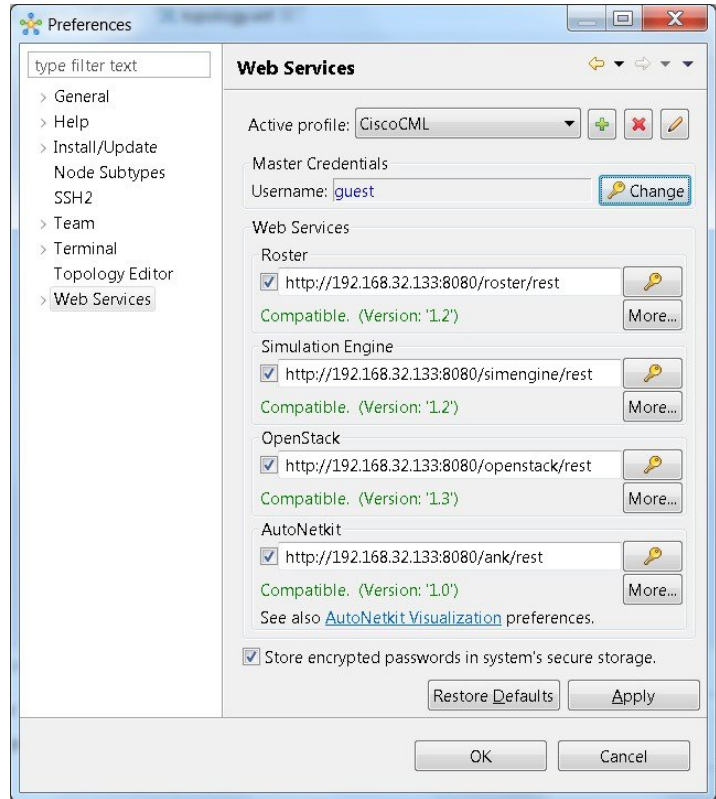
The available operations for this setting are:

Table 32: Web Services Setting Operations

Operation	Description
Active profile	Identifies an active profile that has been defined on the Cisco Modeling Labs client. You can define a new active profile, and edit or delete an existing active profile.
Master Credentials	Specifies a username and password for accessing the Cisco Modeling Labs server. These credentials are provided by the system administrator.

Operation	Description
Web Services	Lists the Web Services needed for the Cisco Modeling Labs server and the Cisco Modeling Labs client to communicate with each other. The Web Services listed are: • **Roster** • **Simulation Engine** • **OpenStack** • **AutoNetkit** **Note** After you have set up your profile with the correct Base URI, each of the Web Services will display a Compatible message in green, indicating that the Cisco Modeling Labs client can communicate with the Cisco Modeling Labs server.
Store encrypted passwords in system's secure storage	Encrypts passwords and stores them locally on the Cisco Modeling Labs client. To change the settings for managing the encrypted passwords, choose **File** > **Preferences** > **General** > **Security** > **Secure Storage**. See the Secure Storage Setting, on page 59 for more information.
Restore Defaults	Restores settings to the initial default state.
Apply	Applies changes.

AutoNetkit Visualization Setting

The AutoNetkit Visualization feature is available only when node configurations are built using the parameters defined in AutoNetkit. If AutoNetkit Visualization is turned off, you cannot get a true representation of your topology.

AutoNetkit Visualization is used to determine how AutoNetkit presents graphical representations of topology-specific attributes, such as nodes, links, and interfaces, during the build phase. The graphical representations can be presented as a physical perspective of a network topology or based on a protocol perspective.

For AutoNetkit Visualization to operate, the Cisco Modeling Labs client must be connected to the Cisco Modeling Labs server. The nodes in the network topology must be set to open AutoNetkit Visualization either automatically, or after receiving a prompt.

To access this setting, choose **File** > **Preferences** > **Web Services** > **AutoNetkit Visualization**.

Figure 41: AutoNetkit Visualization Setting
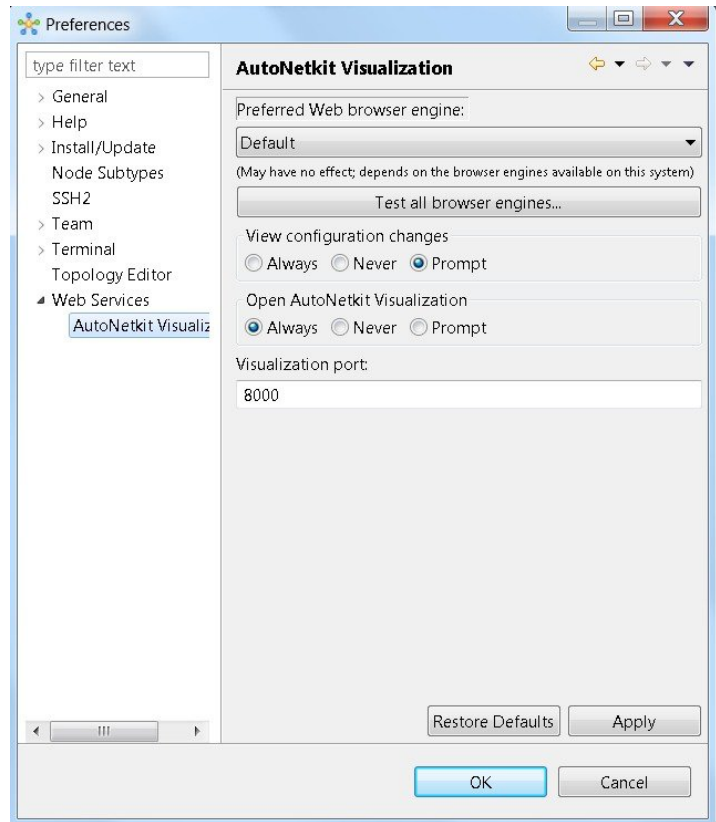
The available operations for this setting are:

Table 33: AutoNetkit Visualization Setting Operations

Operation	Description
Preferred Web browser engine	Specifies the external web browser to use when displaying AutoNetkit Visualization results. Options are: • **Default**—the default system browser • **WebKit**—based on the Safari web browser • **Mozilla Firefox** **Note** You are required to use Mozilla Firefox, Google Chrome, or Apple Safari as your default web browser. Internet Explorer is not supported for AutoNetkit Visualization or for the User Workspace Management interface.
Test all browser engines	Tests the supported browsers that are available.

Operation	Description
View configuration changes	Specifies whether to view configuration changes after a new build is generated. Options are: • **Always** • **Never** • **Prompt** The default value is **Prompt**.
Open AutoNetkit Visualization	Specifies when to open a browser window to display AutoNetkit Visualization. Options are: • **Always** • **Never** • **Prompt** The default value is **Prompt**.
Visualization port	Assigns a port value to the Web Service supporting AutoNetkit Visualization. The default port is 8000. However, the port might need to be changed depending on your network, for example, if a firewall is blocking that port. The port value should be provided by the system administrator. **Note** If an incorrect port value is entered, you will be prompted with an error during the build stage.
Restore Defaults	Restores the default state for the preferred web browser engine and restores the default port value assigned to AutoNetkit Visualization.
Apply	Applies changes .

Web Browser Setting

This setting allows you to add, remove, or edit installed browsers. The selected browser is used by default when web pages are opened in the Cisco Modeling Labs client for AutoNetkit Visualization.

Figure 42: Web Browser Setting
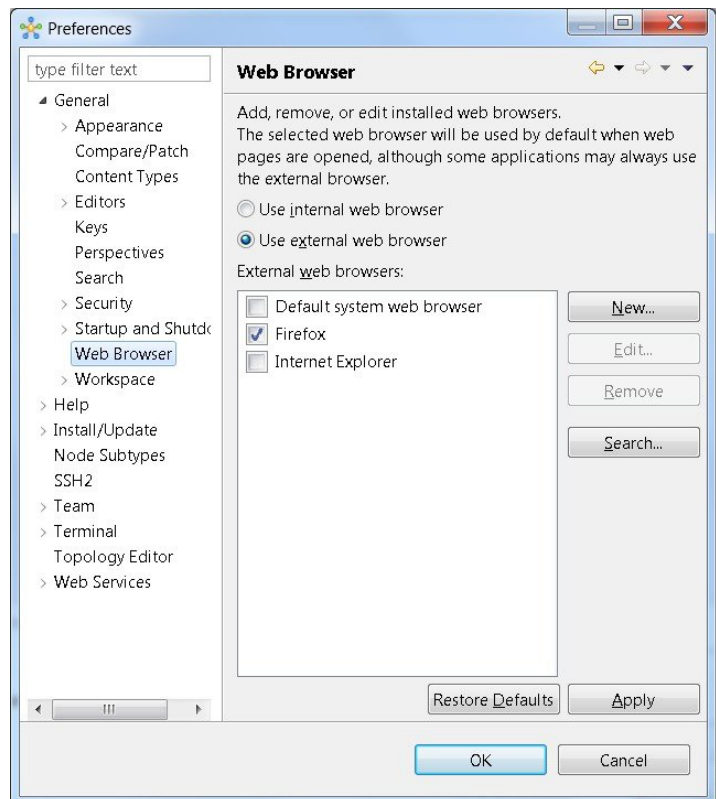
The available operations for this setting are:

*Table 34: Web Browser Setting Operations*

Operation	Description
Use internal web browser	Allows you to use an internal web browser built into the Cisco Modeling Labs client to view AutoNetkit Visualization.
Use external web browser	Allows you to use an external web browser. You can add new browsers, delete or edit existing browsers, or search for new browsers to use. **Note** You are required to use Mozilla Firefox, Google Chrome, or Apple Safari as your default web browser. Internet Explorer is not supported for AutoNetkit Visualization or for the User Workspace Management interface.
Restore Defaults	Restores settings to the initial default state.
Apply	Applies changes.

Secure Storage Setting

This setting configures security preferences and encryption requirements for storing system passwords.

Figure 43: Secure Storage Setting
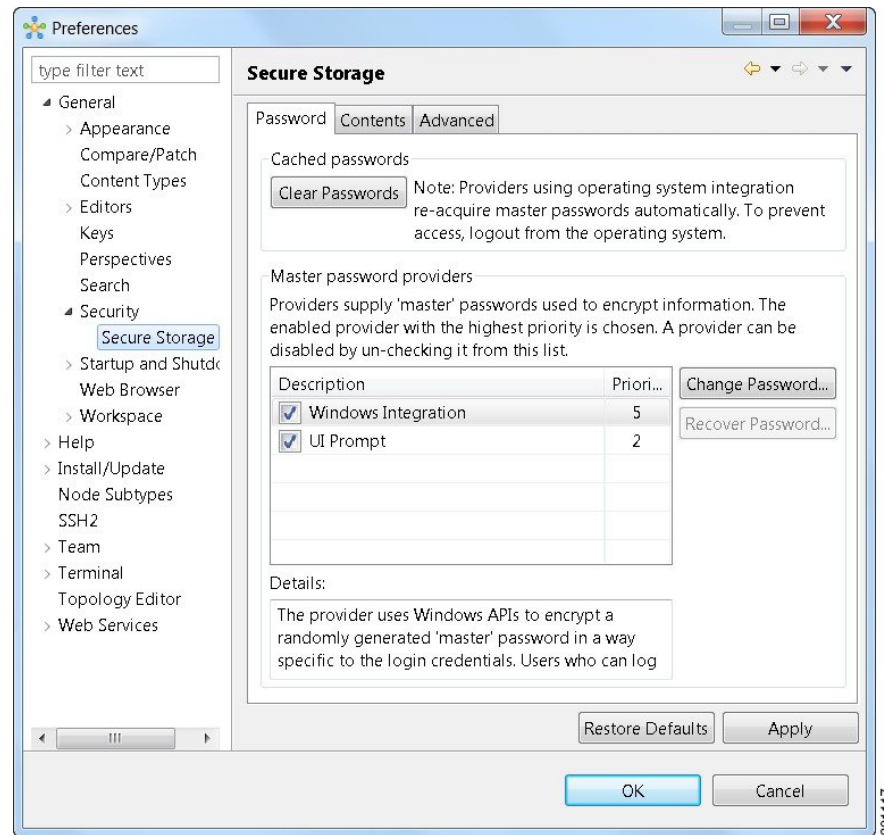
The **Password** tab pools functionality related to the master password life-cycle and password providers.

The available options are:

Table 35: Password Tab Options

Option	Description
Clear Passwords	Clears cached master passwords from memory.

Option	Description
Master password providers	Lists the currently available password providers. By default, the enabled provider with the highest priority is used to encrypt the data added to secure storage. The priority range is from 0 to 10, with 10 being the highest priority. **Note** Data can only be decrypted by the same provider that encrypted the data. By default, all password providers are enabled. Each password provider that has been used at least once will have a master password associated with it. The **Details** text box provides information on the master password providers.
Change Password	Changes the master password of the selected password provider.
Recover Password	Opens the password recovery dialog box. Use this option if you have forgotten the master password and have configured password recovery questions. The button is disabled if the password recovery setup was cancelled when the master password was created. **Note** The answers for the password recovery questions must be entered exactly as they were during the password recovery setup. Answers are case-sensitive, and white space inside answers are relevant.
Restore Defaults	Restores to the initial default state.
Apply	Applies changes.

The **Contents** tab displays contents of the default secure storage. Secure storage is organized as a tree, where nodes represent the context of information and values associated with each node. Selecting a node in the tree displays a table of values associated with that node. Values stored in a nonencrypted form will be displayed; the encrypted values will be shown as *********. At the bottom of this tab, you will find the actual file location used to persist secure storage data. To force the changes to the contents of secure storage to be saved, click **Save**.

To delete stored data in order to recover from an error or to reflect a change in the setup, click **Delete**. This deletes the contents of secure storage. In some cases, other parts of the application may depend on the contents of secure storage that you deleted.

⚠

Caution To avoid unexpected errors, we recommend that you restart the application after secure storage has been deleted.

The **Advanced** tab provides a list of algorithms to further configure secure storage. Changes in the encryption algorithm are applied only to the data stored after a change. If you have already created a secure storage, you must first delete it and then re-create it to use the newly selected encryption algorithm.

Resetting the Secure Storage Password

When the Secure Storage feature is used for the first time, it generates a master password that is used to encrypt the data. In the future, this same master password will be required to retrieve the data from secure storage. If the master password becomes unavailable, the Secure Storage feature provides optional support for password recovery.

Two methods are used to reset the secure storage password.

Method 1

1 From within Cisco Modeling Labs client, choose **File > Preferences > General > Security > Secure Storage**.

2 Click **Change Password**. The **Secure Storage** dialog box appears.

3 Click **Yes**. The **Password Recovery** dialog box appears.

4 Enter details for Question 1 and Question 2, along with answers for both. Take note of the answers you provide, as these are treated as secondary passwords.

5 Click **OK**.

Method 2

If you are unable to access the Cisco Modeling Labs client due to a lost or forgotten secure storage password, complete the following steps:

1 Move to the <user-home>/.eclipse/org.eclipse.equinox.security folder.

2 Delete the file secure_storage.

3 Open Cisco Modeling Labs client and provide details for the secure storage password, when prompted.