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 and 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.

Navigating Within the Cisco Modeling Labs Client

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

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

Figure 1: Menu Bar

File   Edit   View   Configuration   Simulation   Window   Help
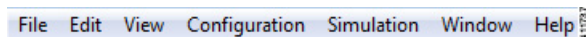
- **Cisco Modeling Labs Client Perspectives**: Identifies the **Design** and **Simulation** perspectives, each of which is associated with an initial set of views and editors in your workbench.

Figure 2: Perspectives

Design   Simulation

  - 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 **Simulation** 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 Views**: Provides alternative presentations of your topology and methods for navigating the information in your workbench.

Figure 3: Views
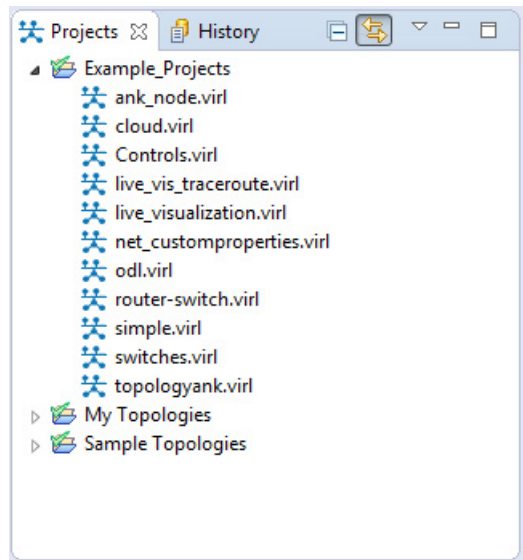
• **Cisco Modeling Labs Client Layout**: Enables you to personalize your workbench, allowing you to rearrange, resize, reset, and move between views.

Menu Bar

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

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.

Generate 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 preselected, 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 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.

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 Topology Editor

The **Topology Editor** is a visual component within the Cisco Modeling Labs client. The **Topology Editor** shows the entire topology (or sites) and it comprises the **Topology Palette** view and the canvas.

The **Topology Editor** allows you to:

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

Cisco Modeling Labs Client Node Icons

The node icons used in the Cisco Modeling Labs client have been updated, as shown in the following figure:

Figure 4: Updated Node Icons

Cisco Modeling Labs Client Unified Editor

This release of the Cisco Modeling Labs client provides the ability to see the interfaces associated with a node, directly in the Design or Simulation perspectives without needing to open a new perspective.
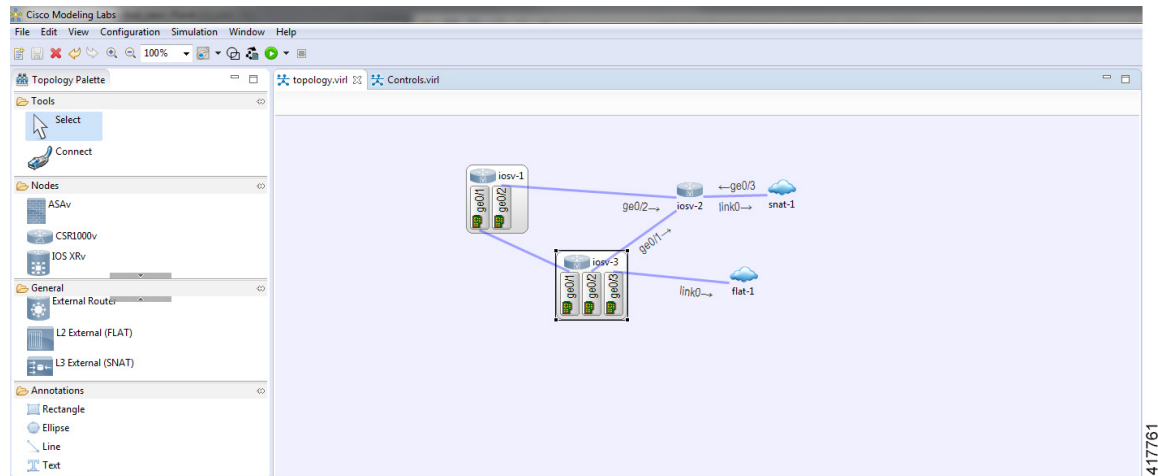
Note   The Node Editor available in previous versions of the Cisco Modeling Labs client has been retired.

Double-clicking a node expands its presentation to show the interfaces and connection mappings.
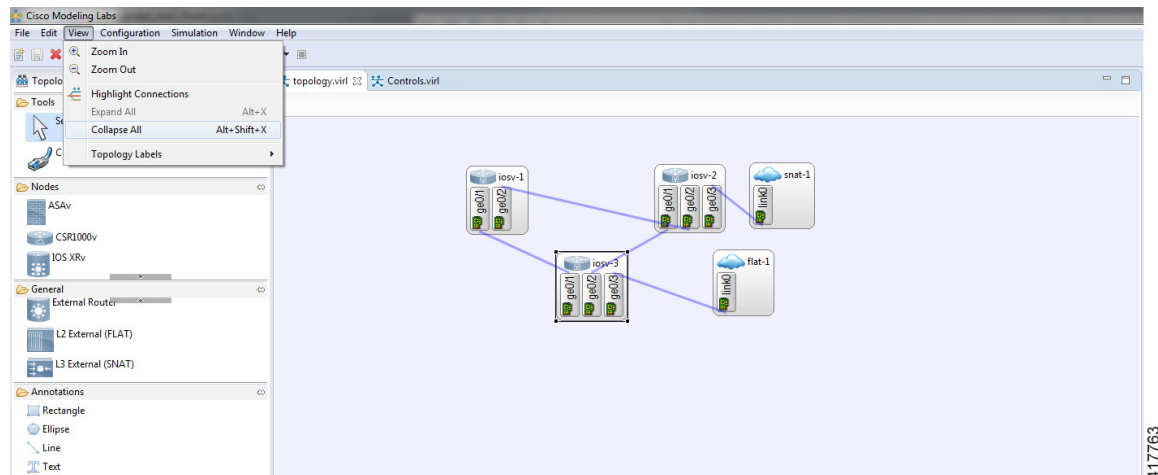
Figure 5: Expanded Nodes with Interfaces Displayed

Double-clicking again hides the node interfaces. The **View/Expand All/Collapse All** menu option can be used to expand and collapse all nodes in your topology.

Figure 6: View All Menu Option

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**: Shows running project simulation(s). The simulation's nodes are listed with their operational state. The Simulation perspective will also present the live-canvas display of the running topology.

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 by choosing **File** > **Preferences** > **General** > **Perspectives**.

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

Table 1: 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 2: 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.)

Customize Perspectives

Cisco Modeling Labs 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 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.
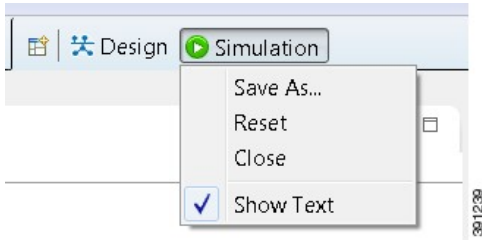
Use this as an initial template from which to create a customized perspective.

Step 2   To add new views, choose **Window** > **Show View** > **Other** and the **Show View** dialog box is displayed listing all views for use.
Views already in use are shown as dimmed in the list.

Step 3   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 4   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.

- **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, right-click the **Design** perspective button, select **Save As**, 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 3: Design Perspective Context Menu Options

Operation	Description
Save As	Saves the customized workbench layout, views, and editors as a new **Design** perspective.

Operation	Description
Reset	Resets the current perspective to display the workbench design that was used when the workbench was first opened. **Note** The **Topology Editor** is not closed when a perspective is reset. You must close it 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	When selected, shows perspective names in the toolbar instead of icons. 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 canvas, **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 4: 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).
Close	Closes the current perspective. (Reopen it using the **Open Perspective** tool.)
Show Text	When selected, shows perspective names in the toolbar instead of icons. When deselected, shows perspective icons in the toolbar instead of names.

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:

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 7: 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 8: Console View Toolbar

Table 5: 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 Console	Opens a new **Console** view.
	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 9: Graph Overview
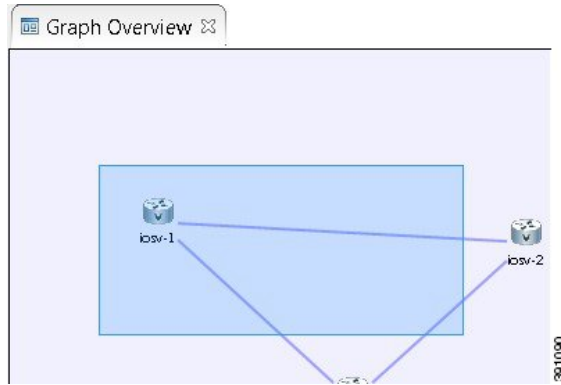
From the **Graph Overview**, click and drag the overlay to pan around your topology. As you drag the overlay, the corresponding content is reflected in the **Topology Editor**.
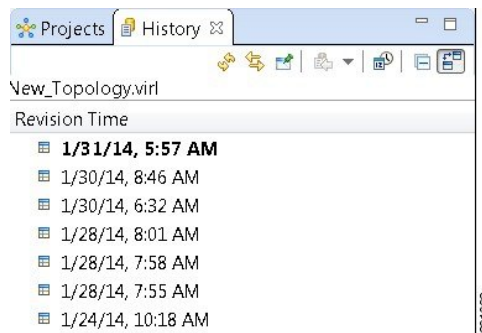
History View

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 10: History View

To view the changed 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; the most recent revision time is 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 11: History View Toolbar

Table 6: 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.
	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.

Topology Palette View

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

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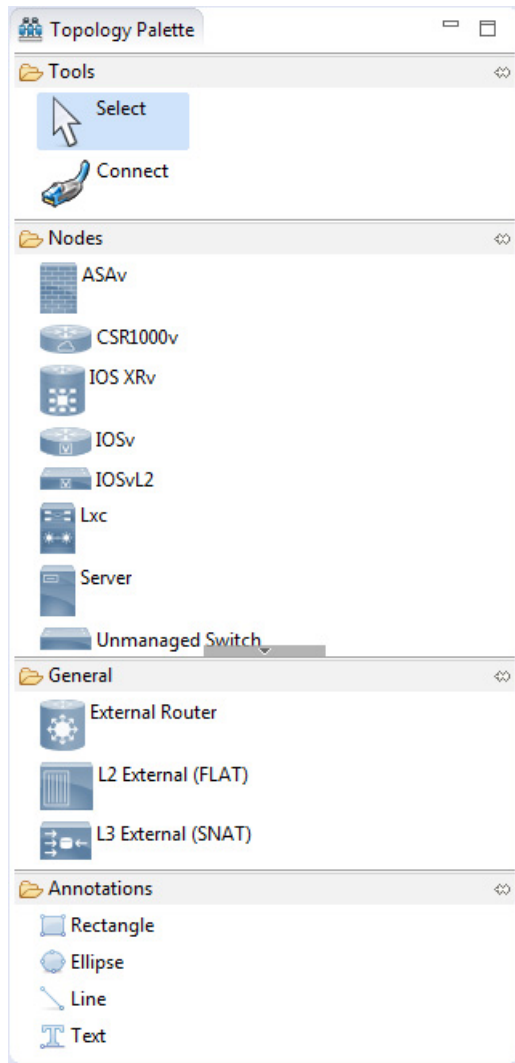
The **Topology Palette** view is divided into the following categories:

- **Tools**: Contains the **Select** and **Connect** tools. The **Select** tool allows you to select nodes, Layer 3 external (SNAT) connections, and Layer 2 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, Release 1.5 includes the following:
    - Cisco IOSv Software Release 15.6(3)T
    - Cisco IOSv Layer 2 Switch Software Release 15.2 (03.2017)
    - Cisco IOS XRv Software Release 6.1.3 CCO
    - Linux server (Ubuntu 16.04.3 Cloud-init)
    - Cisco ASAv Software Release 9.8.2
    - Unmanaged Switch

    > **Note**   Additional node subtypes can be installed separately. See *Release Notes for Cisco Modeling Labs Release 1.5* 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:
    - Layer 3 external connections
    - Layer 2 external connections
    - External routers

- **Annotations**: Contains four types of annotations that you can add to your topologies to provide further information and clarification. These are:
    - Rectangle
    - Ellipse
    - Line
    - Text

The **Topology Palette** view toolbar contains the following tools:
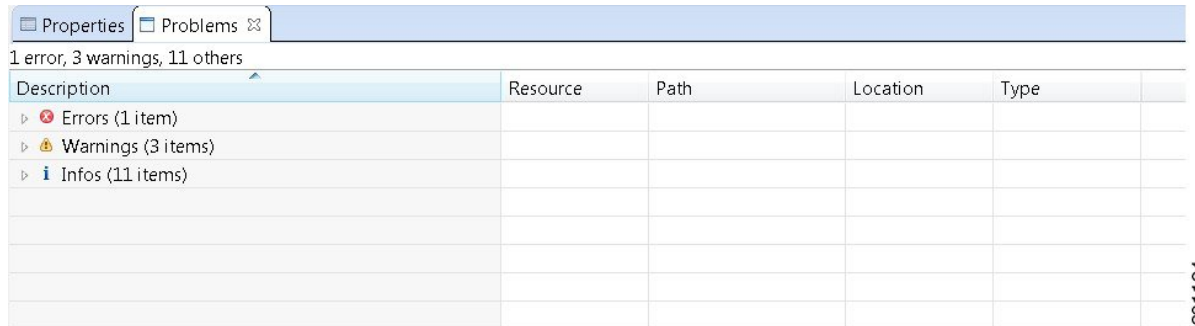
Figure 13: Topology Palette View Toolbar

Table 7: Available Tools

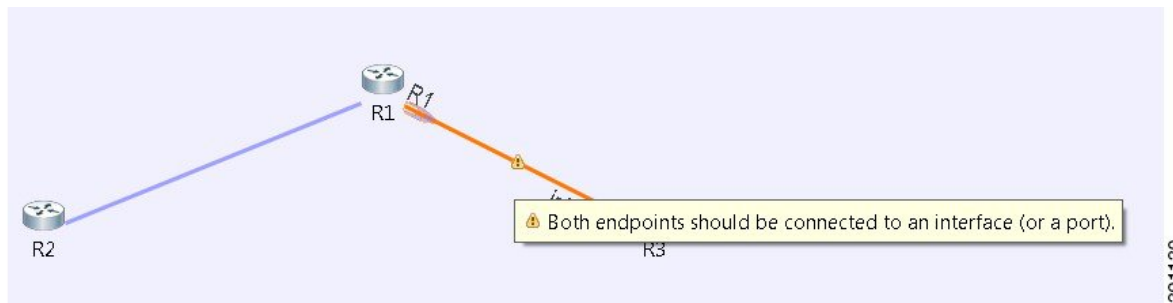| Icon	Tool	Description
	Minimize	Reduces size of **Palette** view.
	Maximize	Increases size of **Palette** view.

Problems View

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 8: Available Tools

Icon	Function	Description

▽	**View Menu**	The **View** menu 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 Problems 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. 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.
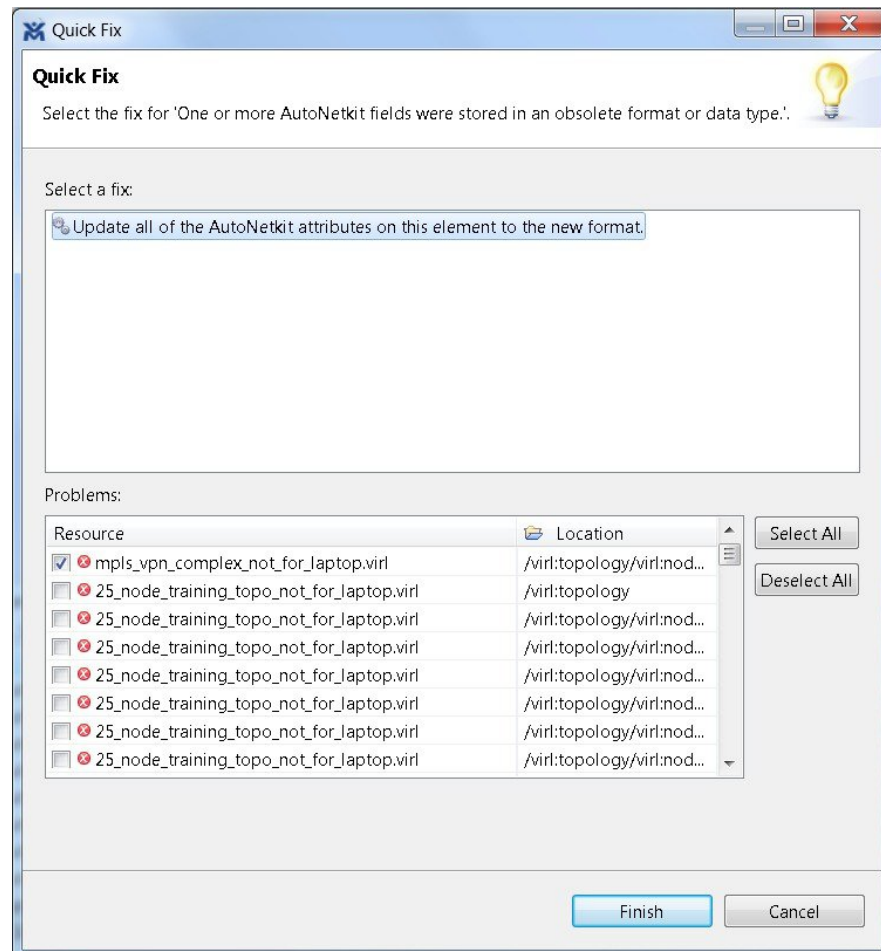
Figure 17: Quick Fix
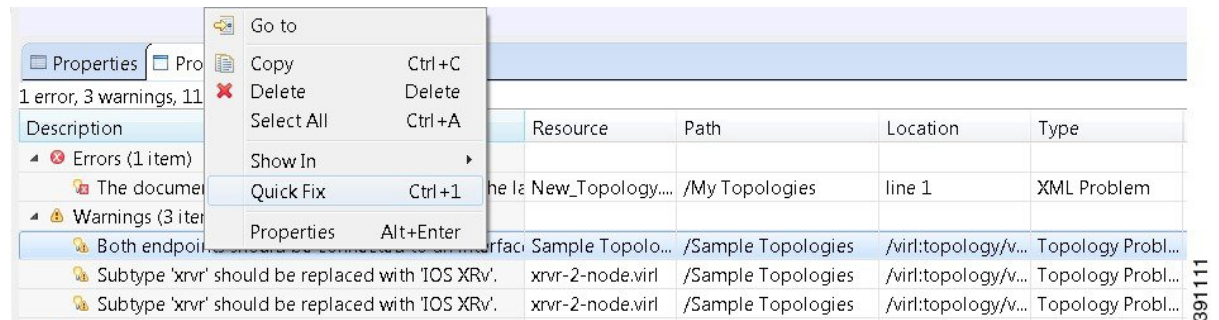
To fix a problem:

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 a 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. In this context menu, 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 9: 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 Menu	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.
	Maximize	Increases the size of the **Projects** view.

The **Projects** view displays several icons on the toolbar:

Table 10: Projects 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.

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

Figure 21: Properties View Toolbar

Table 11: Available Tools

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

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

Figure 22: Node Properties

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

Table 12: Node Properties

Property	Description
Name	Specify a name for the node. **Note** Use only alphanumeric and special 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.

Property	Description
Subtype	Specify a subtype from the list. Cisco Modeling Labs 1.5 includes the following images built into the Cisco Modeling Labs client: • Cisco IOSv Software Release 15.6(3)T • Cisco IOSv Layer 2 Switch Software Release 15.2 (03.2017) • Cisco IOS XRv Software Release 6.1.3 CCO • Linux server (Ubuntu 16.04.3 Cloud-init) • Cisco ASAv Software Release 9.8.2 • Unmanaged Switch Additional Cisco virtual images are available for use. However, they must be installed separately. For a list of supported subtypes, see *Release Notes for Cisco Modeling Labs 1.5*. **Note** When changing a node subtype after it is initially configured using AutoNetkit, for example, changing a Cisco IOSv subtype to a Cisco IOS XRv subtype, you need to revalidate 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.
Static Serial Ports	Lists the following ports: • **(C)onsole Port:** An optional static TCP port number assigned to the node's console port. The number must be unique and in the range 4000..32767. • **(A)uxillary Port:** An optional static TCP port number assigned to the node's auxillary port. The number must be unique and in the range 4000..32767. • **(M)onitor Port:** An optional static TCP port number assigned to the node's monitor port. The number must be unique and in the range 4000..32767. • **(E)xtra Port:** An optional static TCP port number assigned to the node's extra port. The number must be unique and in the range 4000..32767.
VM Image	Specify a 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.
Other | Management Interface Static IPv4 Address	Specify the IPv4 address to be assigned to the management interface of the node. The address must be in the IPv4 subnet assigned to the Flat or Flat1 network.

The property **Exclude Node from Simulation Launch** is set on a per-node basis. When it is enabled for a node, the node is not launched when the simulation is launched. However, the node can later be started and configured and automatically join a running simulation.

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 **Build Initial Configurations**.

Note
Any preexisting configuration for a node is overwritten when you choose **Build Initial Configurations** from the toolbar. Uncheck this **Auto-generate the configuration based on these attributes** check box if you do not want the router configuration for a node updated by AutoNetkit.

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

Table 13: AutoNetkit Properties

Property	Fields	Description
General	**ASN**	Specify the autonomous system number, which is used to infer IGP and BGP. This can be any valid integer.
IGP	**IGP**	Configure an internal routing protocol. Options are: • **Not specified** • **OSPF** • **ISIS** • **EIGRP** • **RIP-V2** The default value is **Not specified**.
	OSPF Area	Configure an OSPF area. The default value is **0**.
iBGP	**iBGP Role**	Configure an iBGP role from the list and use it to create an iBGP topology. Options are: • **Not specified** • **Disabled** • **Peer** • **RR** (route reflector) • **HRR**(hierarchical route reflector) • **RRC**(route reflector client) The default value is **Peer**.
	RR Cluster	Specify the RRC as a name or number. Should be an alphanumeric string.

Property	Fields	Description
	HRR Cluster	Specify the HRR cluster. Should be an alphanumeric string.
Custom Configuration	Global	In this section, users can specify their own configuration text for inclusion in the appropriate section of the node configuration. **Note** The following fields do not apply to external routers. Note too that text entered is not syntactically checked, so ensure that the text is valid.
	Physical Interfaces	Specify a custom configuration for the physical interfaces.
	Loopback Zero	Specify a custom configuration for loopback zero.
	OSPF	Specify a custom configuration for OSPF.
	IS-IS	Specify a custom configuration for IS-IS.
	EIGRP	Specify a custom configuration for EIGRP.
	RIP-V2	Specify a custom configuration for RIPv2.
	BGP	Specify a custom configuration for BGP.
MPLS	VRF Name	Specify an MPLS VPN name for VRF MPLS VPNs.
	LDP	Enable Cisco MPLS Label Distribution Protocol. Options are: • **Not specified** • **True** • **False** The default value is **False**.
	Enable MPLS TE	Enable Cisco MPLS Traffic Engineering. Options are: • **Not specified** • **True** • **False** The default value is **False**.
External BGP	IPv4 Address	**Note** This property only applies to external routers. Specify the IPv4 address of the remote router. Enter a valid IP address in the correct format.
	IPv6 Address	Specify an IPv6 address of the remote router. Enter a valid IP address in the correct format.
	Remote ASN	Specify the AS number of the remote router. This is used when trying to establish a BGP connection to a remote device. The value range is 1 to 65535.

Property	Fields	Description
	MD5 Password	Specify the MD5 password to use to secure the BGP session to the remote router.
	Multihop	Enable BGP multihop. When the remote router is directly adjacent (Layer 3 adjacent), this field is set to **False**; otherwise it is set to **True**. Options are: • **Not specified** • **True** • **False** The default value is **True**.
External L2TPv3	**Remote Loopback IPv4 Address**	Specify a remote loopback IPv4 address.
	Local Endpoint IPv4 Address	Specify a local endpoint IPv4 address.
	Local Endpoint IPv4 Netmask	Specify a local endpoint IPv4 netmask.
	PseudoWire ID	Specify a pseudowire ID.
GRE Tunnel	**IPv4 Tunnel Enabled**	Enable IPv4 GRE tunneling. Options are: • **Not specified** • **True** • **False** The default value is **False**.
	Tunnel IPv4 Address	Specify a tunnel IPv4 address to use, which is the IP address of the far-end node terminating the GRE tunnel itself.
	Tunnel IPv4 Netmask	Specify a tunnel IPv4 netmask to use.
	IPv6 Tunnel Enabled	Enable IPv6 GRE tunneling. Options are: • **Not specified** • **True** • **False** The default value is **False**.

Property	Fields	Description
	Tunnel IPv6 Address	Specify a tunnel IPv6 address to use, which is the IP address of the far-end node terminating the GRE tunnel itself.
	Tunnel IPv6 Netmask	Specify a tunnel IPv6 netmask to use.
OpenDayLight (ODL)	**OpenDayLight (ODL) Management Group**	Cisco IOS XRv devices set with the **ODL Management Group** attribute must be paired with an External Router entity which is configured with the matching ODL Management Group attribute along with an ODL External Server IP address. The ODL server may be running on your Cisco Modeling Labs server or another location and does not need to part of the Cisco Modeling Labs simulation itself. Connectivity between the simulation and server must be provided.

Information displayed under the **Configuration** tab depends on whether 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 with the **Key**, **Value**, and **Type** attributes.

Cisco Modeling Labs Client Node Menu Options

Cisco Modeling Labs has previously provided users with a series of extensions that could be applied to Cisco Modeling Labs topologies in order to control aspects such as the Mgmt-IP address assigned to a node or the static_ip address to be applied to a data interface.

The Cisco Modeling Labs client provides node type appropriate menu options, allowing users to easily set these values, without the need to the use the extensions function.

Figure 23: IOSv Node Menu Options

Figure 24: Flat Node Menu Options

Figure 25: SNAT Node Menu Options

The following table provides an expanded list of the various extensions.

Extension Name	Use	Example	Applies to	Note
host_network (string)	Defines which Flat network to use.	host_network=flat1	Topology or Flat connector	If undefined, 'flat' is used.
static_ip (string)	Defines a static IP assignment of an object.	static_ip=172.16.1.200 static_ip=10.254.0.222	Flat and SNAT objects and nodes for OOB management IP	Single IP for flat networks, IP address pair for SNAT mappings. Admin rights required.
bound_port (string)	References a pre-configured Neutron port to use.	bound_port=my_static_port	Flat /SNAT connector or virtual machine instance	Ports must be defined via the User Workspace Management interface.
vlan (integer)	VLAN number for IP insertion.	vlan=100	Flat/SNAT attached to Cisco OS node instance	Inserts 'Float IP' into interface that has VLAN 100.
subinterface (string)	Subinterface name for IP insertion.	subinterface=.100	Flat/SNAT attached to Cisco OS node instance	Inserts 'Float IP' into interface that ends with .100.
initial config (string)	Create file on node disk.	initial config script.txt=line 1 line 2 line 3	Cisco IOSv/IOSvL2 node instance	File name is part of key, which in the example used is script.txt.
lxc.host.static_ip (string)	Defines which IP the LXC jumphost should use.	lxc.host.static_ip=172.16.1.254	Topology with the 'Use an LXC Management node' parameter set to True. (management_lxc=true)	If undefined, a pool IP is used.

Extension Name	Use	Example	Applies to	Note
lxc.host.bound_port (string)	References a pre-configured Neutron port to be used by the LXC jumphost.	lxc.host.bound_port=lxcport	Topology with the 'Use an LXC Management node' parameter set to True. (management_lxc=true)	The Neutron port must be on a Flat network.
lxc.host.tcp_port (string)	Defines the TCP port on the Cisco Modeling Labs host that the LXC jumphost is available on.	lxc.host.tcp_port=12345	Topology with the 'Use an LXC Management node' parameter set to True. (management_lxc=true)	

Topology Properties

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

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

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

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

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

Interface Properties

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

Under the **Extensions** tab, all the extensions used to generate the configuration are listed 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 the **Connection** tab, 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, from the toolbar choose **Edit** > **Search**. The **Search** dialog box is displayed. Enter criteria for your search and click **Search**. The **Search** view displays the results of the 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, which 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 26: Search View
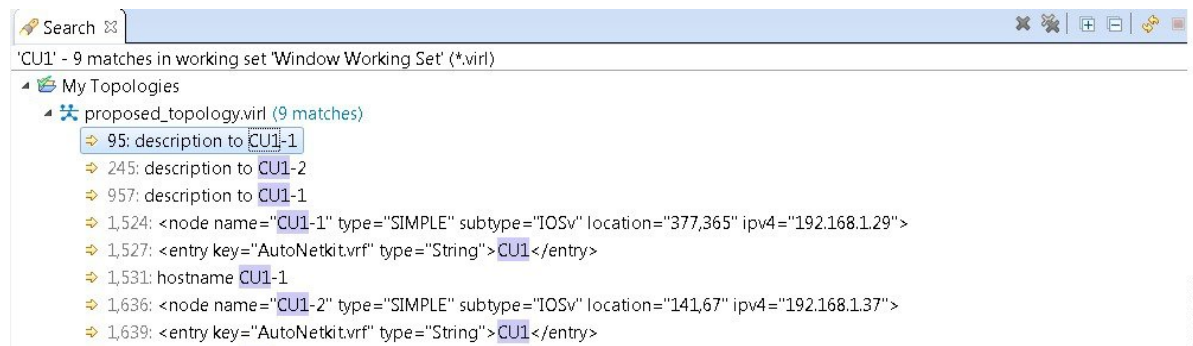
To use the search functionality, perform the following tasks:

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 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 27: Search View Toolbar

Table 14: 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.
	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 and selects a previous search from the drop-down menu to repeat a previous search. 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 while the pinned view remains unchanged. This allows for a comparison of 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 searches 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

Possible simulation states are:

Table 15: Simulation States

Possible node states are:

Table 16: Node States

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

Figure 28: Simulations View Toolbar

Table 17: Available Tools

Icon	Function	Description
	Refresh the List	Refreshes the list of simulations displayed in the **Simulations** view.
	Minimize	Reduces the size of the **Simulations** view.
	Maximize	Increases the size of the **Simulations** view.

Topology Options

Node Options

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 29: Terminal View

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

Figure 30: Terminal View Toolbar

Table 18: Available Tools

Icon	Function	Description
	Disconnect	Disconnects the terminal connection to the node.

Icon	Function	Description
	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.
	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.

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.

The available operations for this setting are:

Secure Storage Setting

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

The **Password** tab pools functionality related to the master password life cycle and password providers.

The available options are:

Table 19: Password Tab Options

Option	Description
Clear Passwords	Clears cached master passwords from memory.
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 recreate 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 password for the secure storage feature.

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 in both Question fields and provide answers for both questions. 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 password for the secure storage feature, 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 to provide details for the password for the secure storage feature when prompted.

Node Subtypes Setting

The available operations for this setting are:

Fetch 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 finish.

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

Packet Capture Setting

This setting allows you to check for suitable applications that can open .pcap files when you have downloaded packet captures. The .pcap files can be set to either open automatically or after receiving a prompt.

The available operations for this setting are:

Table 20: Packet Capture Setting Operations

Operation	Description
Check for Applications Capable of Opening *.pcap Files	Check the checkbox box to have the system automatically check for a suitable application to open the .pcap files.
Open Downloaded *.pcap Files	Select the applicable option: **Always**, **Never** or **Prompt** for opening downloaded .pcap files.
Restore Defaults	Restore the settings to their initial default state.
Apply	Apply any changes made.

Simulation Launch Setting

This setting allows you to include the file's path in the system-generated Simulation IDs. You can set the system to either open the Simulation perspective automatically or after receiving a prompt.

The available operations for this setting are:

Table 21: Simulation Launch Setting Operations

Operation	Description
Include the File's Path in Sytem generated Simulation IDs.	Check this box if you want to include the file's path in the system-generated Simulation IDs.
Switch to the Simulation Perspective after a Simulation is Launched.	Select the applicable option: **Always**, **Never** or **Prompt** for switching to the Simulation perspective after a simulation is launched.
Restore Defaults	Restore the settings to their initial default state.
Apply	Apply any changes made.

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** and **Simulation** perspectives. 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 WebSocket to a VM, the terminal opens internally, not externally as specified.

The available operations for this setting are:

Setting Up an External Terminal

This section outlines the steps involved in setting up an external terminal on Windows and OS X.

Windows

Under **File > Preferences > Terminal > Cisco Terminal**, update the terminal settings for connection to an external program. For example, for a PuTTY installation, configure the putty.exe binary file as the terminal program.

Note The complete PATH must be specified.

Table 22: PuTTY Command-Line Options

Command-Line Option	Description
ssh	Opens an SSH connection
telnet	Opens a Telnet connection
%h	Specifies the host to connect to (the Cisco Modeling Labs client will not allow an external program to be set without having **%h** in the command line)
%p	Specifies the host to connect to (the Cisco Modeling Labs client will not allow an external program to be set without having **%p** in the command line)

For an external terminal on Windows, the commands to enter are:

- Telnet command: `"C:\Program Files (x86)\PuTTY\putty.exe" -telnet %h %p`
- SSH command: `"C:\Program Files (x86)\PuTTY\putty.exe" -ssh %h %p`

Note The double quotes (" ") must enclose the PATH to allow the use of spaces within the PATH. Select the **Use external terminal application (specified below)** radio button to use an external terminal.

The following table lists other terminal programs that can be used.

Table 23: Additional Terminal Programs

Terminal Program	Connection Type	Command to Use
Xshell	**Telnet**	`"C:\Program Files (x86)\NetSarang\Xshell 4\xshell.exe" -url telnet://%h:%p -newtab %t`
SecureCRT	**Telnet**	`"C:\Program Files\VanDyke Software\SecureCRT\SecureCRT.exe" /T /TELNET %h %p` **Note** The **/T** option ensures SecureCRT creates a tab for new sessions instead of opening a new window. Ensure to validate the path of the binary.

OS X

The procedure is more complex on OS X since command-line parameters are not as easy to use on this platform. Two additional components are required. These are:

- AppleScript: A program used to start applications and interact with them; some examples are open windows, start new sessions, paste keyboard input into a session, and so on.

- /usr/bin/osascript: A built-in OS X command-line utility used to execute AppleScript and other OSA language scripts. This is configured in the Cisco Modeling Labs client; it is essentially the glue between the Cisco Modeling Labs client and the terminal application.

The following table lists iTerm and the built-in Terminal.app programs that can be used on OS X.

Table 24: Additional Terminal Programs for OS X

Terminal Program	Connection Type	Command to Use
iTerm	**Telnet**	`/usr/bin/osascript /Users/your-user-id/iterm.scpt telnet %h %p %t`
	SSH	`/usr/bin/osascript /Users/your-user-id/iterm.scpt telnet %h %p %t`
Terminal.app	**Telnet**	`/usr/bin/osascript /Users/your-user-id/iterm.scpt ssh %h %p %t`
	SSH	`/usr/bin/osascript /Users/your-user-id/terminal.scpt ssh %h %p %t`

The following are AppleScript scripts that are used to set up an external terminal.

iTerm

```
on run argv
    tell application "iTerm"
        activate
        if current terminal exists then
            set t to current terminal
```

```
      else
          set t to (make new terminal)
      end if

      tell t
          launch session "Default Session"
          tell the current session
              write text "/usr/bin/" & item 1 of argv & " " & item 2 of argv & " " & item
 3 of argv
              set name to item 4 of argv
          end tell
      end tell
    end tell
end run
```

Terminal.app

```
on run argv
    tell application "Terminal"
        activate
        -- open a new Tab - there is no method
        tell application "System Events"
            keystroke "t" using {command down}
        end tell
        repeat with win in windows
            try
                if get frontmost of win is true then
                    set cmd to "/usr/bin/" & item 1 of argv & " " & item 2 of argv & " " &
 item 3 of argv
                    do script cmd in (selected tab of win)
                    set custom title of (selected tab of win) to item 4 of argv
                end if
            end try
        end repeat
    end tell
end run
```

Topology Editor Setting

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

The available operations for this setting are:

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 **Unauthorized** in red. This message relates to the **Master Credentials** field, which must be set before the Cisco Modeling Labs client can communicate with the Cisco Modeling Labs server.

The available operations for this setting are:

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**.

The available operations for this setting are: