Cisco Modeling Labs 1.0 User Guide

First Published: August 04, 2014

Americas Headquarters
Cisco Systems, Inc.
170 West Tasman Drive
San Jose, CA 95134-1706
USA
http://www.cisco.com
Tel: 408 526-4000
800 553-NETS (6387)
Fax: 408 527-0883

Text Part Number: OL-32056-01

CONTENTS

Preface

Document Conventions

This document uses the following conventions:

Convention	Description
^ or Ctrl	Both the ^ symbol and Ctrl represent the Control (Ctrl) key on a keyboard. For example, the key combination **^D** or **Ctrl-D** means that you hold down the Control key while you press the D key. (Keys are indicated in capital letters but are not case sensitive.)
bold font	Commands and keywords and user-entered text appear in **bold** font.
Italic font	Document titles, new or emphasized terms, and arguments for which you supply values are in *italic* font.
`Courier` font	Terminal sessions and information the system displays appear in `courier` font.
`Bold Courier` font	**`Bold Courier`** font indicates text that the user must enter.
[x]	Elements in square brackets are optional.
...	An ellipsis (three consecutive nonbolded periods without spaces) after a syntax element indicates that the element can be repeated.
|	A vertical line, called a pipe, indicates a choice within a set of keywords or arguments.
[x | y]	Optional alternative keywords are grouped in brackets and separated by vertical bars.

Convention	Description
{x | y}	Required alternative keywords are grouped in braces and separated by vertical bars.
[x {y | z}]	Nested set of square brackets or braces indicate optional or required choices within optional or required elements. Braces and a vertical bar within square brackets indicate a required choice within an optional element.
string	A nonquoted set of characters. Do not use quotation marks around the string or the string will include the quotation marks.
< >	Nonprinting characters such as passwords are in angle brackets.
[ ]	Default responses to system prompts are in square brackets.
!, #	An exclamation point (!) or a pound sign (#) at the beginning of a line of code indicates a comment line.

Reader Alert Conventions

This document may use the following conventions for reader alerts:

Note Means *reader take note*. Notes contain helpful suggestions or references to material not covered in the manual.

Tip Means *the following information will help you solve a problem.*

Caution Means *reader be careful*. In this situation, you might do something that could result in equipment damage or loss of data.

Timesaver Means *the described action saves time.* You can save time by performing the action described in the paragraph.

Warning IMPORTANT SAFETY INSTRUCTIONS

This warning symbol means danger. You are in a situation that could cause bodily injury. Before you work on any equipment, be aware of the hazards involved with electrical circuitry and be familiar with standard practices for preventing accidents. Use the statement number provided at the end of each warning to locate its translation in the translated safety warnings that accompanied this device. Statement 1071

SAVE THESE INSTRUCTIONS

Related Documentation

Note Before installing Cisco Modeling Labs 1.0, refer to the Cisco Modeling Labs release notes.

These documents provide complete information on Cisco Modeling Labs 1.0:

- Cisco Modeling Labs 1.0 Corporate Edition System Administrator Installation Guide
- Cisco Modeling Labs 1.0 Corporate Edition Client Installation Guide
- Cisco Modeling Labs 1.0 User Guide
- Release Notes for Cisco Modeling Labs 1.0

Obtaining Documentation and Submitting a Service Request

For information on obtaining documentation, submitting a service request, and gathering additional information, see the monthly *What's New in Cisco Product Documentation*, which also lists all new and revised Cisco technical documentation, at:

http://www.cisco.com/c/en/us/td/docs/general/whatsnew/whatsnew.html

Subscribe to the *What's New in Cisco Product Documentation* as a Really Simple Syndication (RSS) feed and set content to be delivered directly to your desktop using a reader application. The RSS feeds are a free service and Cisco currently supports RSS version 2.0.

CHAPTER 1

Overview of Cisco Modeling Labs

Cisco Modeling Labs

Cisco Modeling Labs 1.0 is a scalable and extensible software platform that enables operators, engineers, network designers, and architects to design Cisco-based networks and run simulations using virtual versions of selected Cisco operating systems. Cisco Modeling Labs comprises the Cisco Modeling Labs server and the Cisco Modeling Labs client. Together, they provide a sandbox environment that facilitates the design, configuration, visualization, and simulation of network topologies quickly and efficiently.

- **Cisco Modeling Labs server—**A shared resource containing the capability to initiate topologies using installed virtual images.
- **Cisco Modeling Labs client—**A point-and-click GUI that simplifies topology creation and initial device configurations along with continuous updates, and permits access to the Cisco Modeling Labs server functionality.
- **Virtual Images—**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. See the *Release Notes for Cisco Modeling Labs 1.0* for the most up-to-date list of supported virtual images.

  An Ubuntu 14.04 server has been tested with Cisco Modeling Labs. 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**.

Together, the virtual images and Ubuntu server provide a powerful platform for creating, modifying, and testing the virtual network scenarios.

Cisco Modeling Labs Server Components

The Cisco Modeling Labs server is a Linux distribution that is bundled within the VMware Open Virtual Appliance (OVA) file for VMware ESXi. The bundle includes all the supporting files. The Cisco Modeling Labs server is a shared resource used by end users to run backend functions, such as router bootstrap configurations, spinning up routers to operate with designated operating systems, and modifying and testing configurations.

Cisco Modeling Labs 1.0 comprises a framework of components. The main components are:

- **OpenStack—**An open-source platform for creating and managing large groups of virtual servers in a cloud-computing configuration. It is used for node control, management, and networking.
- **AutoNetkit—**An automated configuration engine that uses templates to provide working router configurations based on user-supplied and default parameters for each virtual machine.
- **Services Topology Director—**Generates OpenStack calls for the creation of virtual machines and links based on the XML topology definition created by the Cisco Modeling Labs client. Additionally, it provides the bootstrap configuration, which can be auto-generated, generated manually, or imported.

Cisco Modeling Labs Client

The Cisco Modeling Labs client is a cross-platform user interface for creating and editing network designs, and simulating those network topologies on the Cisco Modeling Labs server. The Cisco Modeling Labs client offers the following benefits:

- The ability to quickly create and edit complex network topologies in a sandbox, using a graphical point-and-click editor.
- Access to the build, visualization, and launch functions available in the Cisco Modeling Labs server.

The Cisco Modeling Labs client enables you to interact directly with your running simulations from the user interface. The Cisco Modeling Labs client also provides the functionality to generate default router configurations before launching the topology simulation.

For further information on the Cisco Modeling Labs client, see Using the Cisco Modeling Labs Client Overview, on page 7.

Virtual Images

Cisco Modeling Labs 1.0 includes the Cisco IOSv 15.4(2)T virtual image. The operation and syntax used with the virtual images are the same as a physical platform running the same software.

See the *Release Notes for the Cisco Modeling Labs 1.0* for more information on Cisco IOSv virtual software supported features.

Cisco Modeling Labs Component Requirements

The following tables identify the minimum requirements for installing and operating the Cisco Modeling Labs 1.0 Corporate Edition. For additional information, see the *Cisco Modeling Labs 1.0 Corporate Edition Client Installation Guide* and the *Cisco Modeling Labs 1.0 Corporate Edition System Administrator Installation Guide*. These guides provide detailed information on platform requirements for installing and operating Cisco Modeling Labs 1.0 Corporate Edition for system administrators and end users.

Table 1: Server Hardware Requirements

Requirement	Description
Small to Medium	Server with a capacity to run 30-40 nodes
Memory (RAM)	128 GB
Disk Space	1 TB minimum
Processors	16 CPU cores
Large	Server with a capacity to run 40-100 nodes
Memory (RAM)	256 GB
Disk Space	1 TB minimum
Processors	40 CPU cores

Table 2: Server Software Requirements

Requirement	Description
VMware vSphere	One of the following: • Release 5.0 with VMware ESXi • Release 5.1 with VMware ESXi • Release 5.5 with VMware ESXi
Browser	One of the following: • Google Chrome Version 33.0 or later • Internet Explorer 10.0 or later • Mozilla Firefox 28.0 or later • Safari 7.0 or later **Note** Internet Explorer is not supported for use with the AutoNetkit Visualization functionality and the User Workspace Management interface.

Table 3: Client Hardware Requirements

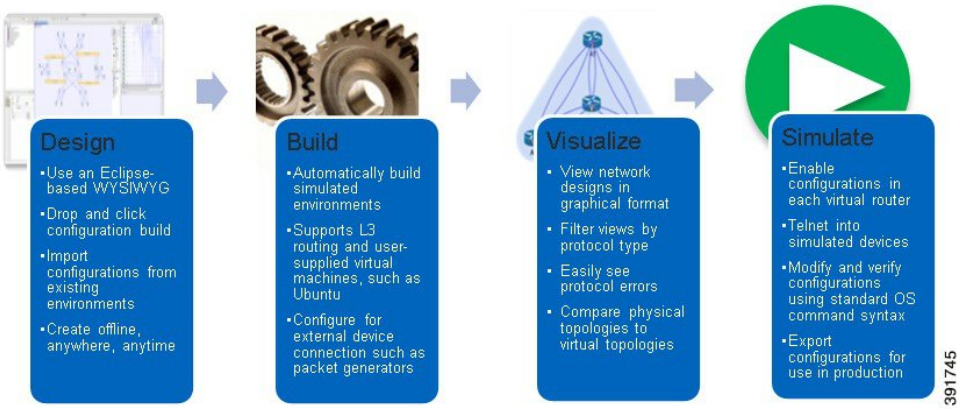
Requirement	Description
Operating System	Either of the following: • Microsoft Windows ◦ Windows 7 or ◦ Windows 8 • Apple Mac OS X ◦ Mac OS X 10.8 or later
Memory (RAM)	500 MB
Disk Space	150 MB

Table 4: Client Software Requirements

Requirement	Description
Java Runtime Environment (JRE)	For Windows and Mac OS X, either of the following: • Version 6 • Version 7
Browser	One of the following: • Google Chrome Version 33.0 or later • Internet Explorer 10.0 or later • Mozilla Firefox 28.0 or later • Apple Safari 7.0 or later **Note** Internet Explorer is not supported for use with the AutoNetkit Visualization functionality and the User Workspace Management interface.

Cisco Modeling Labs Framework

Figure 1: Cisco Modeling Labs Framework



Cisco Modeling Labs 1.0 includes numerous features that enable you to create and simulate small and large network designs. This user guide is organized in a task-based format where the main features are grouped into four sections referred to as phases.

The following items describe each phase which should help you determine which section to refer to when using this guide:

1. **Design—**This phase includes the tasks for creating a network topology. You use a blank canvas to create topologies from scratch or import existing network topologies. You can also view your topologies on a geographic map and adjust where and how interfaces are used on each device.
2. **Build—**This phase includes the tasks associated with configuring routers, external connections, and servers, creating the required configurations, setting up interfaces, IP addressing, and routing protocols for the virtual routers. There are several ways to create these configurations. You can use the AutoNetKit functionality to set up the initial configuration or you can input your own configuration details. Whatever configurations you create in this phase will be the configurations that the Cisco Modeling Labs server will use when it initiates the node simulations.
3. **Visualization—**This phase is optional and operates only if you use AutoNetKit to create your configurations during the Build phase. It includes the tasks related to running visualization scenarios of your network design and configuration. It provides visual views of your topology whereby you can see how the nodes will interact with each other in specific circumstances including physical set up as well as with specific routing protocols such as ISIS and OSPF. It also supports MPLS and BGP.
4. **Simulation—**This phase includes the tasks for initiating the nodes and making them active. Once the nodes are operational, you can connect to the consoles using Telnet or SSH, as you would connect to a router console. You can run connectivity tests and modify configurations. This is where the power of the product is realized since you can modify and test configurations as if you were on actual physical devices. In this phase, you can also save your configurations and extract them for sharing with others or saving them and using them as reference when configuring the production network.

CHAPTER 2

Using the Cisco Modeling Labs Client

Using the Cisco Modeling Labs Client Overview

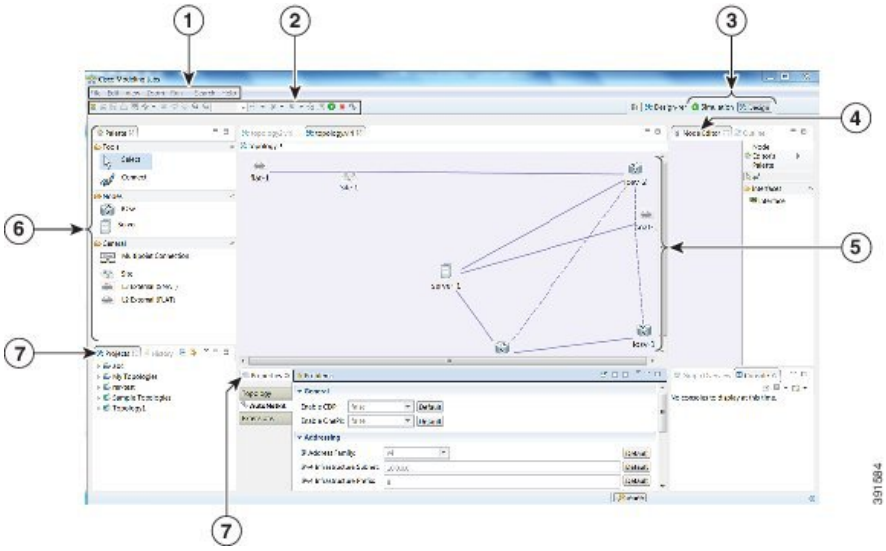
The Cisco Modeling Labs client is a cross-platform, point-and-click GUI that simplifies topology creation, initial-device configurations, and permits access to the Cisco Modeling Labs server. You can interact directly with your running simulations from this GUI. Additionally, the Cisco Modeling Labs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

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 2: 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 5: 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, on page 69 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 12 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 12 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 52 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 13 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 16
• Cisco Modeling Labs Client Perspectives, on page 17
• Cisco Modeling Labs Client Views, on page 21

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 3: Topology Editor

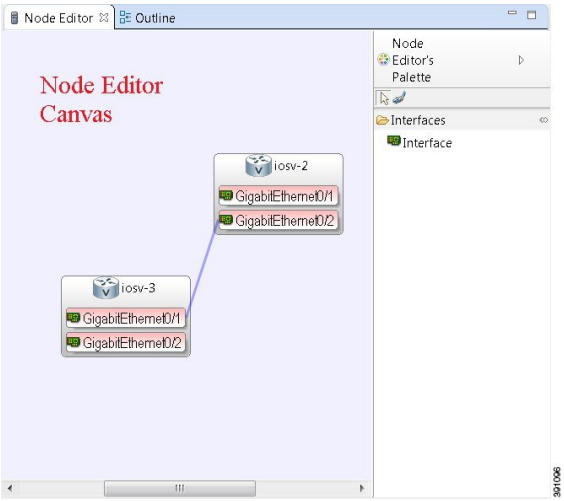
Node Editor

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 4: 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.

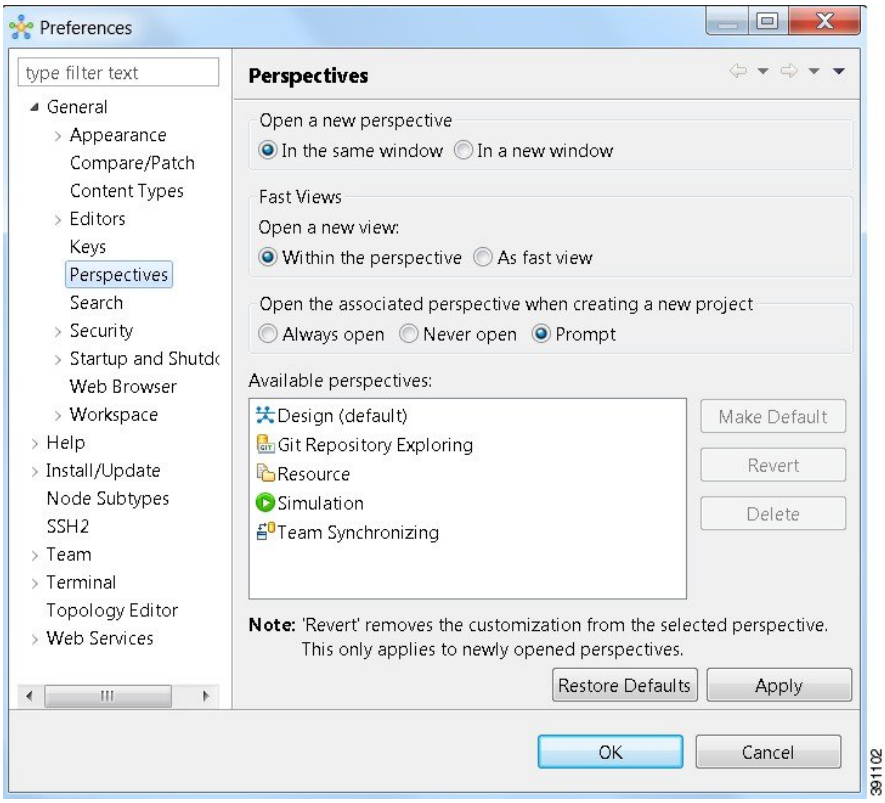
Note The **Design** and **Simulation** perspectives that are built into the Cisco Modeling Labs client cannot be deleted.

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 5: Perspective Preferences



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

Table 6: 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 7: 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.)

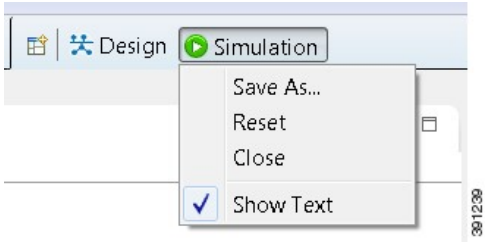
Customizing Perspectives

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 8: 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 9: 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 10: Commonly Used Views in the Cisco Modeling Labs Client

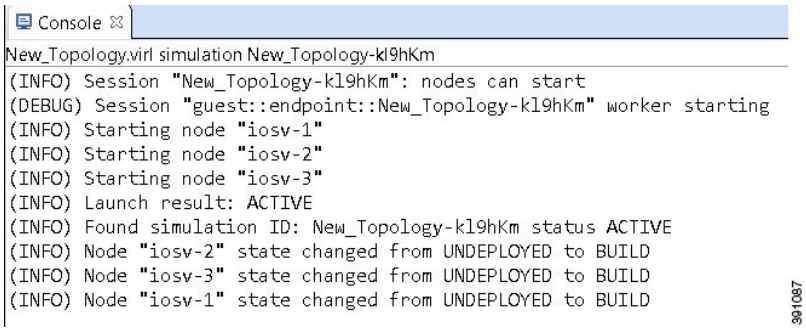
View Name	Perspective Where Used	Description
Console view	Design and Simulation	**Design—**Displays messages from AutoNetkit when it is used to generate router configurations. **Simulation—**Displays message streams from the Cisco Modeling Labs server after a simulation is launched. For more information, see Console View, on page 23
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 24

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 24
Outline view	Design	Shows a list of the nodes in the topology and their interface connections. For more information, see Outline View, on page 26
Palette view	Design	Allows you to add devices and interface connections to your topology. For more information, see Palette View, on page 27
Problems view	Design	Displays errors, warnings, and other information that was detected within the **Topology Editor**. For more information, see Problems View, on page 29
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 34
Properties view	Design	Displays names and properties of nodes and interfaces. For more information, see Properties View, on page 35
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 44
Simulations view	Simulation	Displays information on all running simulations. For more information, see Simulations View, on page 46
Terminal view	Simulation	Displays console information when you connect to a node using Telnet. For more information, see Terminal View, on page 51

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 6: Console View



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

Figure 7: Console View Toolbar

Table 11: 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.

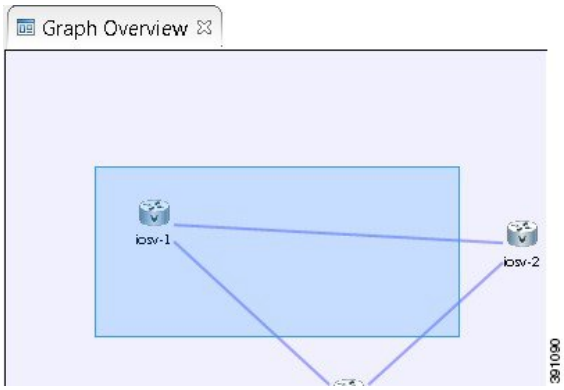
Note When several simulations are running, use the toolbar button **Display Selected Console** to toggle between the **Console** views for the different simulations.

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 8: 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**.

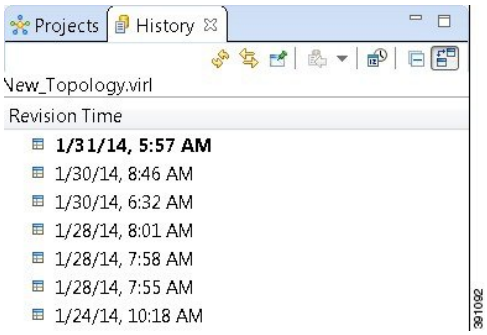
History View

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 9: 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 10: History View Toolbar

Table 12: Available Tools

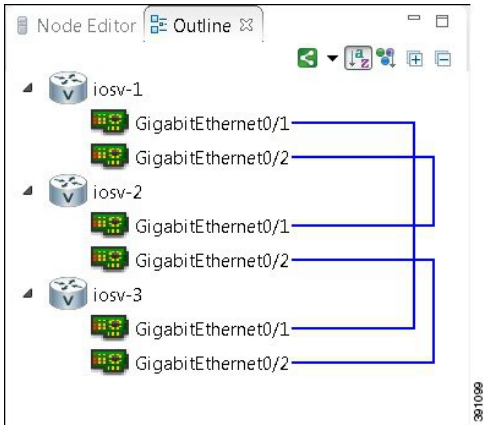
Icon	Function	Description
	Refresh	Refreshes the contents of the view, retrieving the latest history information for a file from the system.
	Link with Editor and Selection	Toggles when the **History** view selection is linked to the active editor. When this option is selected, changing the active editor 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 11: Outline View



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

Figure 12: Outline View Toolbar

Table 13: Available Tools

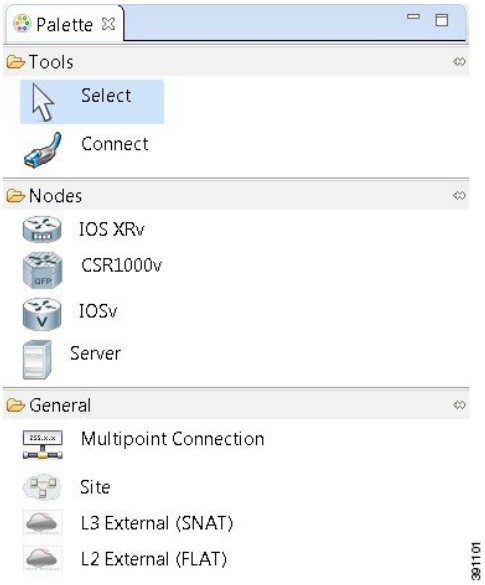
Icon	Function	Description
	**Change Display Mode of Connection**	Allows you to choose various options when displaying connections: • **Hide Connections—**No connections are shown. **Show Selected Connections—**Displays only the connections that are selected in the **Topology Editor**. • **Show Intra-Topology 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 13: 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 14: Palette View Toolbar

Table 14: Available Tools

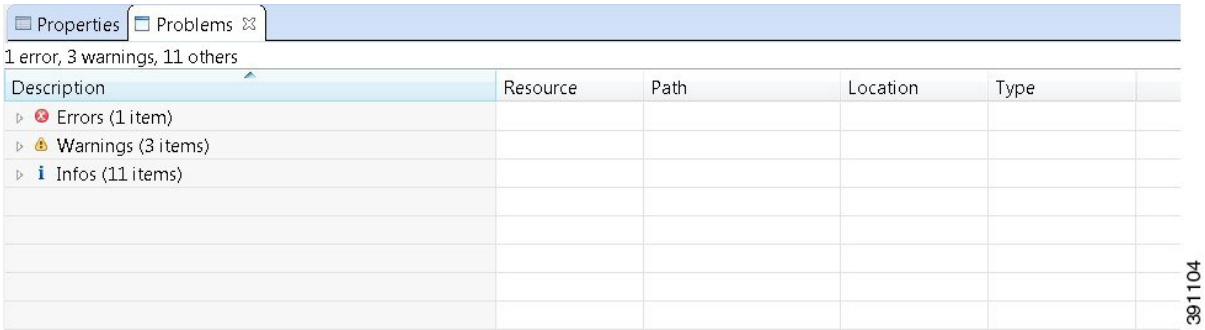
Icon	Tool	Description
	**Minimize**	Reduces size of **Palette** view.
	**Maximize**	Increases size of **Palette** view.

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.

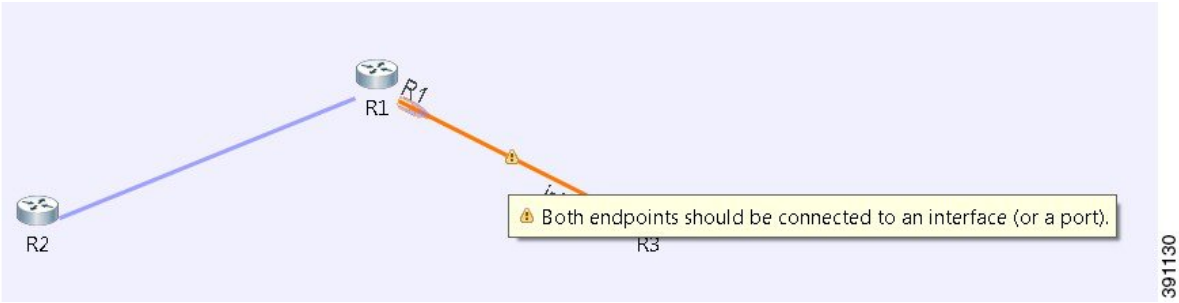
Figure 15: Problems View



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 16: Problem Example



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

Figure 17: Problems View Toolbar

Table 15: Available Tools

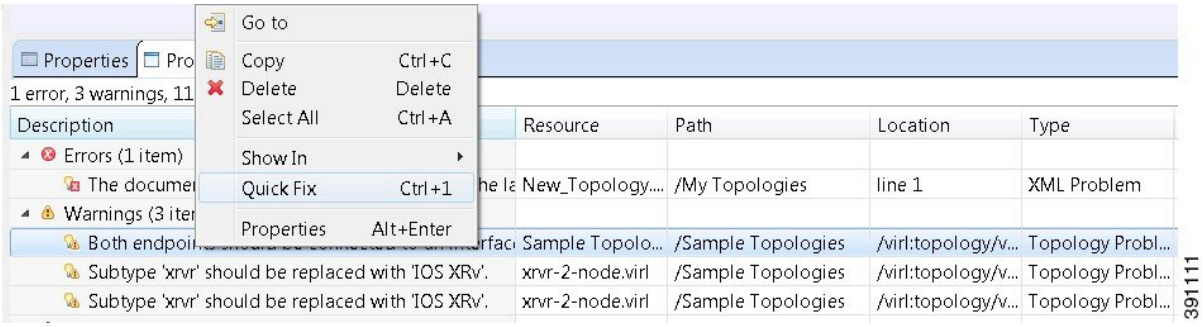
Icon	Function	Description
	View Menu	The **Problems** view tool has the following options: • **Show—**Displays errors and warnings. • **Group By—**Groups problems under the headings **Type**, **Severity**, and **None**. The default is **Severity**. • **Sort By—**Sorts problems under the following headings: ◦ **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 18: Quick Fix

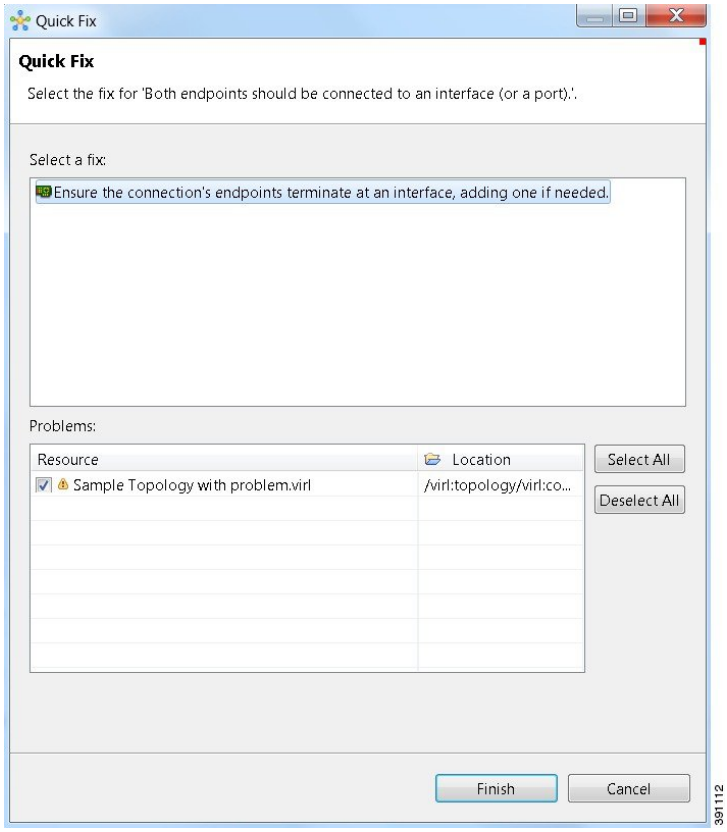


To fix a problem:

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

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

Figure 19: 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 20: 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 21: Projects View Toolbar

Table 16: 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:

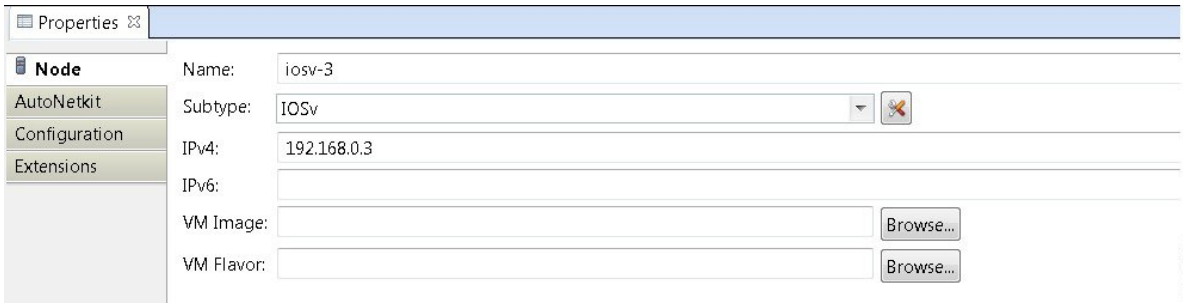
Table 17: Project View Icons

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 22: Properties View



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

Figure 23: Properties View Toolbar

Table 18: 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**.

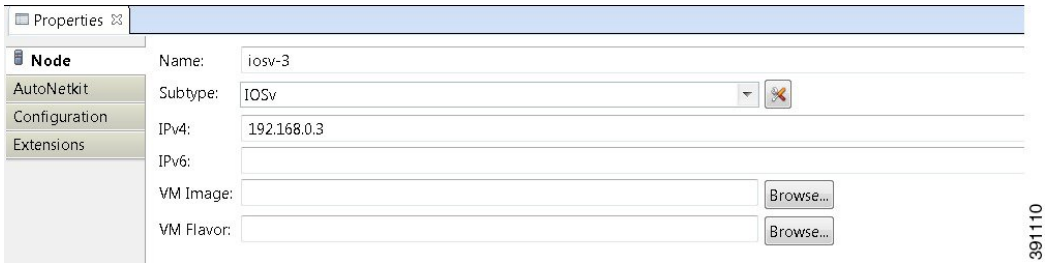
The properties in the **Properties** view are discussed in the following sections:

- Node Properties, on page 37
- Topology Properties, on page 39

Node Properties

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

Figure 24: Node Properties



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

Table 19: 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 20: 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.

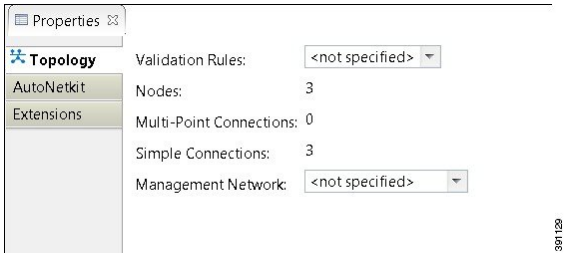
- When checked, AutoNetkit generates the configuration and displays it under the node's **Configuration** tab.
- When unchecked, no configuration information is created for the node. You must configure the node manually, or cut and paste the existing configuration information into this area.

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 25: Topology Properties



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

Table 21: 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 22: 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 23: 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 24: 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 25: 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

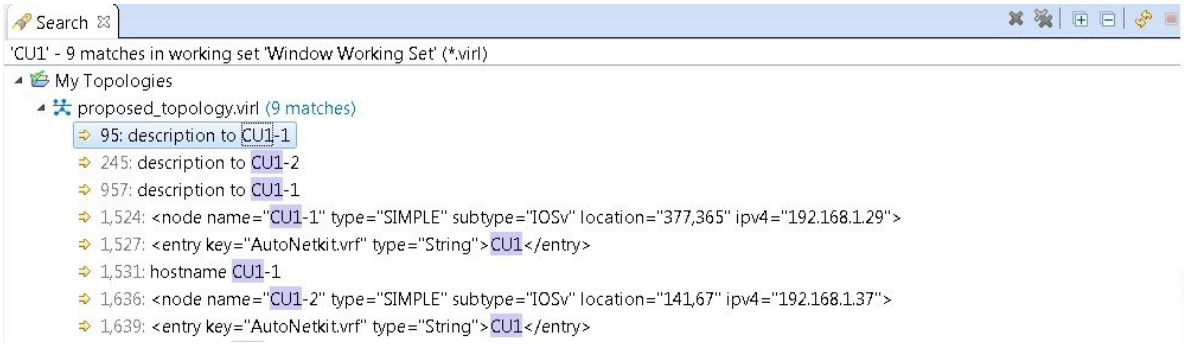
To search for text string and files, click **Search** on the toolbar. The **Search** view displays the results of a search. A file search can be based on text strings, regular expressions, and patterns, in addition to whole words and case-sensitive characters. The scope of a file search can encompass 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 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 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 27: Search View Toolbar

Table 26: 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.

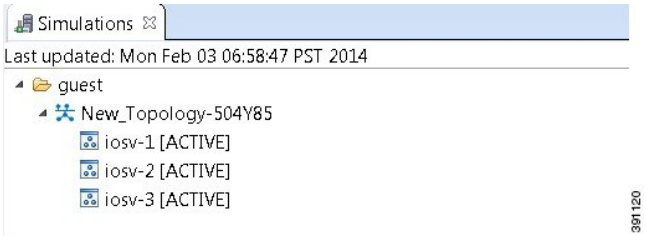
Simulations View

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 28: Simulations View



Possible simulation states are:

Table 27: 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 28: 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 29: Simulations View Toolbar

Table 29: 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 30: Topology Options

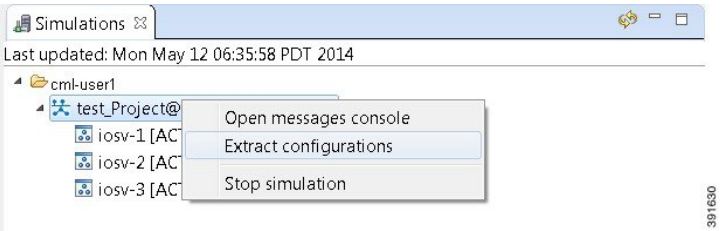


Table 30: Topology Options

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, on page 101 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 31: Node Options

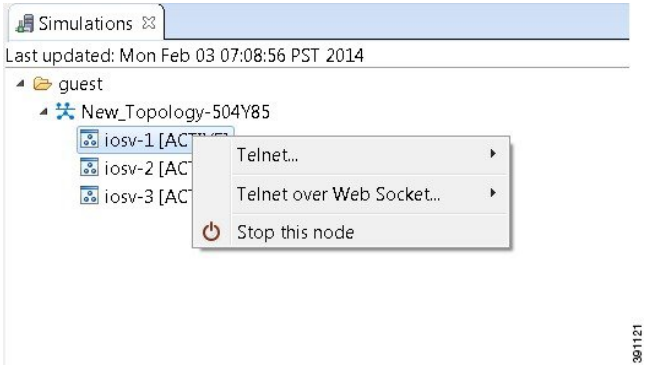


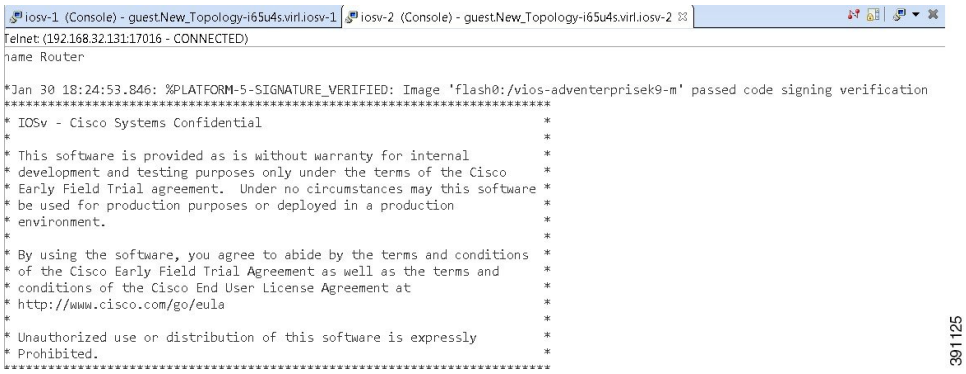
Table 31: Node Options

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, on page 109 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 32: Terminal View



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

Figure 33: Terminal View Toolbar

Table 32: 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**:

- Node Subtypes Setting, on page 52
- Terminal Setting, on page 56
- Topology Editor Setting, on page 58
- Web Services Setting, on page 59
- AutoNetkit Visualization Setting, on page 61
- Web Browser Setting, on page 64
- Secure Storage Setting, on page 65

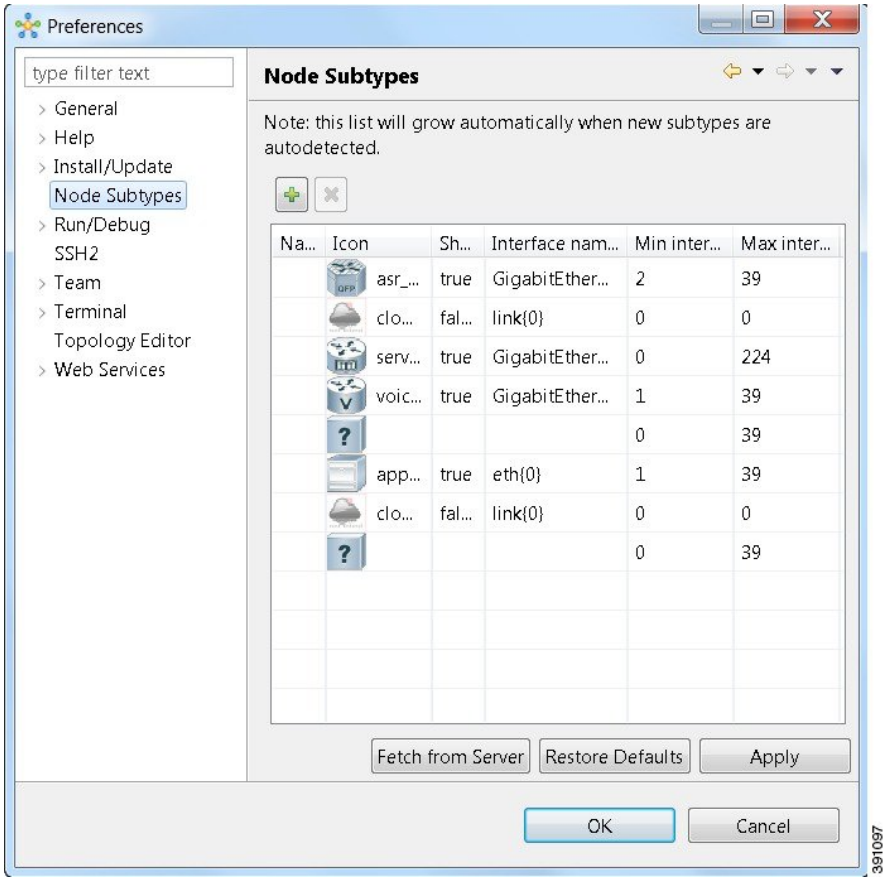
These are discussed in the following sections.

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 34: Node Subtypes Setting



The available operations for this setting are:

Table 33: 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 54.
	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 55.
	Restore Defaults	Reverts to the original list of subtypes.
	Apply	Applies changes.

Adding a New Node Subtype to the List

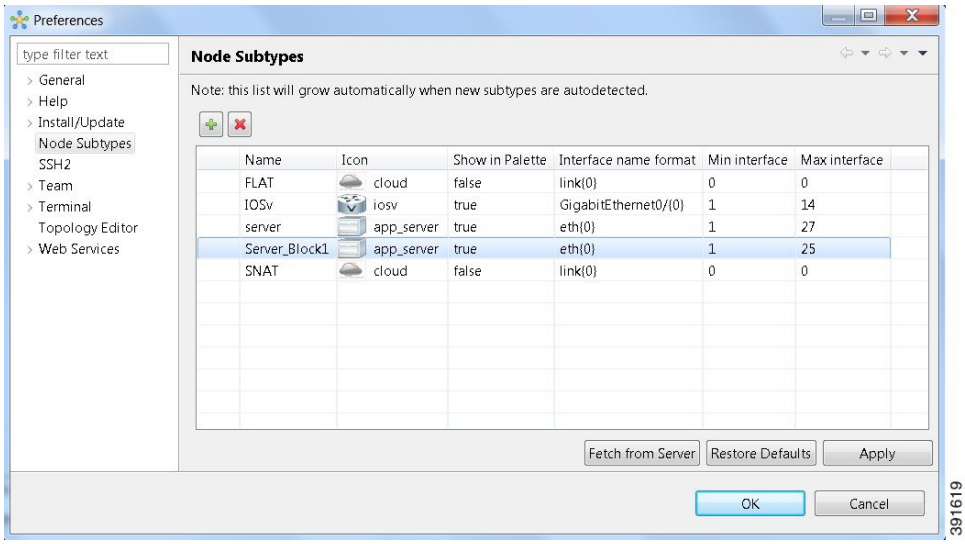
To add a new node subtype to the node subtype list, perform the following tasks:

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 35: 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 36: New Node Subtype Visible in the Palette View

Fetching Node Subtypes from the Cisco Modeling Labs Server

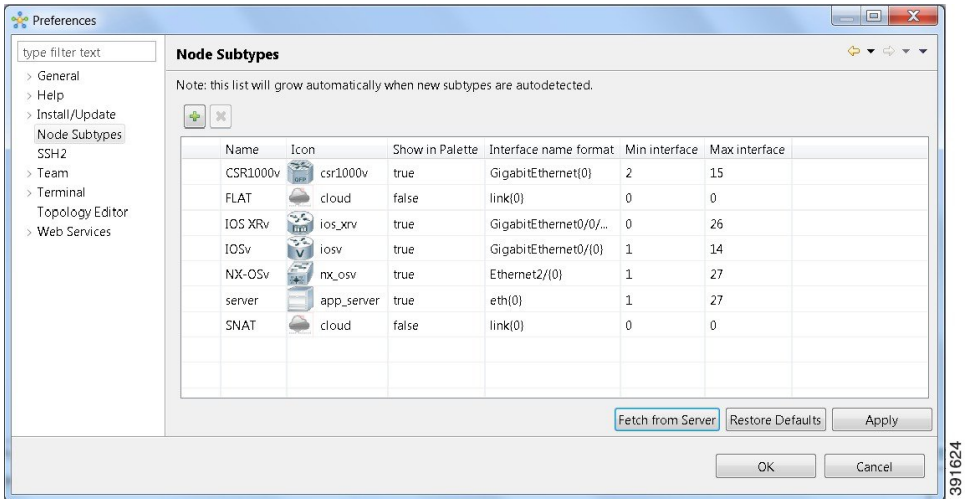
To fetch new node subtypes from the Cisco Modeling Labs server, perform the following tasks:

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.

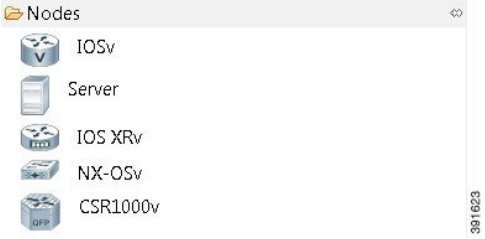
Figure 37: Fetch Nodes Subtypes from Server



Step 4 Click **OK** to finish.

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

Figure 38: 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.

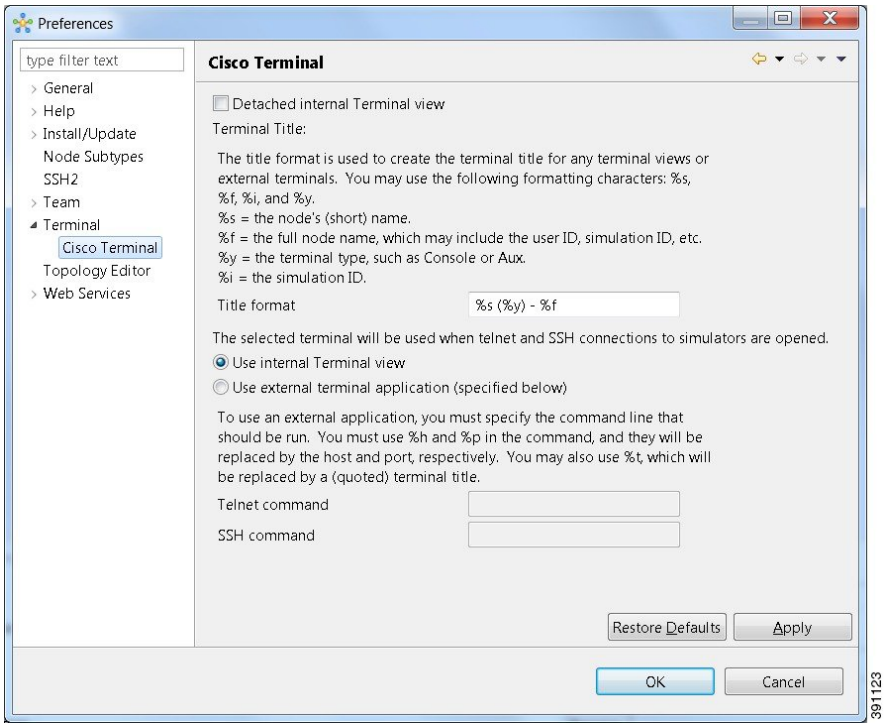
Terminal Setting

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 39: Terminal Setting



The available operations for this setting are:

Table 34: Terminal Setting Operations

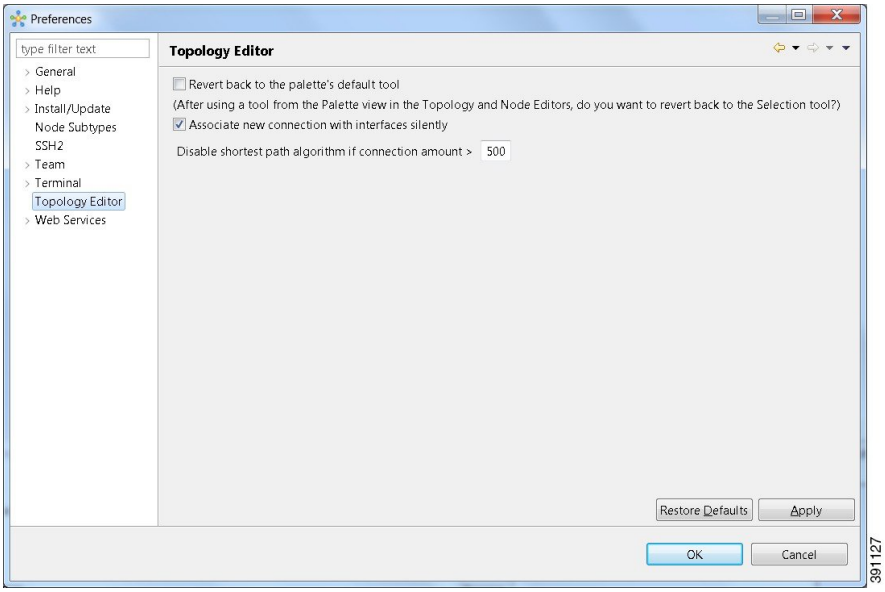
Operation	Description
Detached internal Terminal view	Launches the **Terminal** view in a separate window.
Title format	Allows you to add a **Terminal** view title using the required formatting characters.
Use internal Terminal view	Uses the Cisco Modeling Labs internal **Terminal** view.

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 40: Topology Editor Setting



The available operations for this setting are:

Table 35: 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.

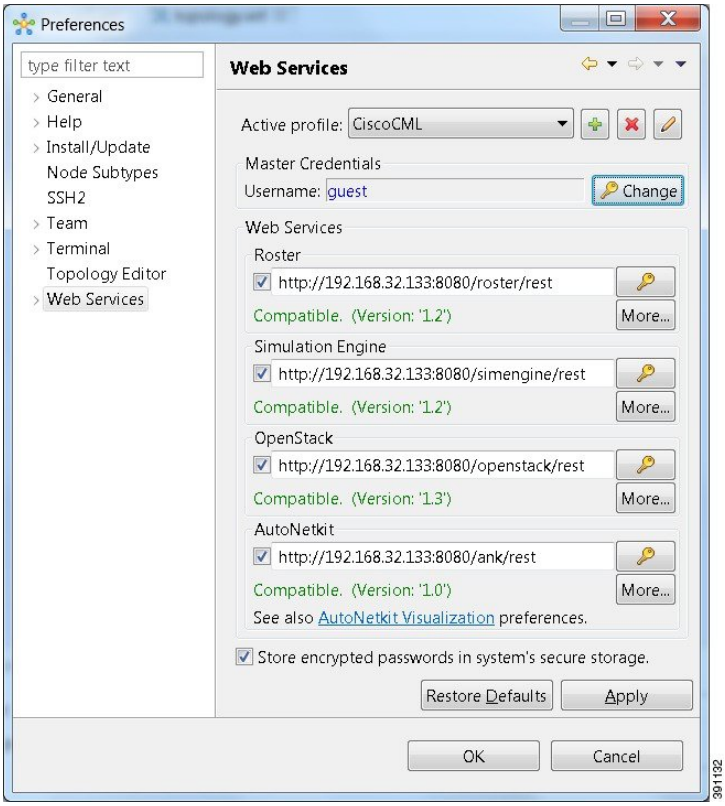
Operation	Description
Associate new connection with interfaces silently	Enables the following: • When selected, interfaces are automatically assigned to connections between nodes. • When deselected, you are presented with a list of available interfaces.
Disable shortest path algorithm if connection amount > (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 41: Web Services Setting



The available operations for this setting are:

Table 36: 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 65 for more information.
Restore Defaults	Restores settings to the initial default state.
Apply	Applies changes.

AutoNetkit Visualization Setting

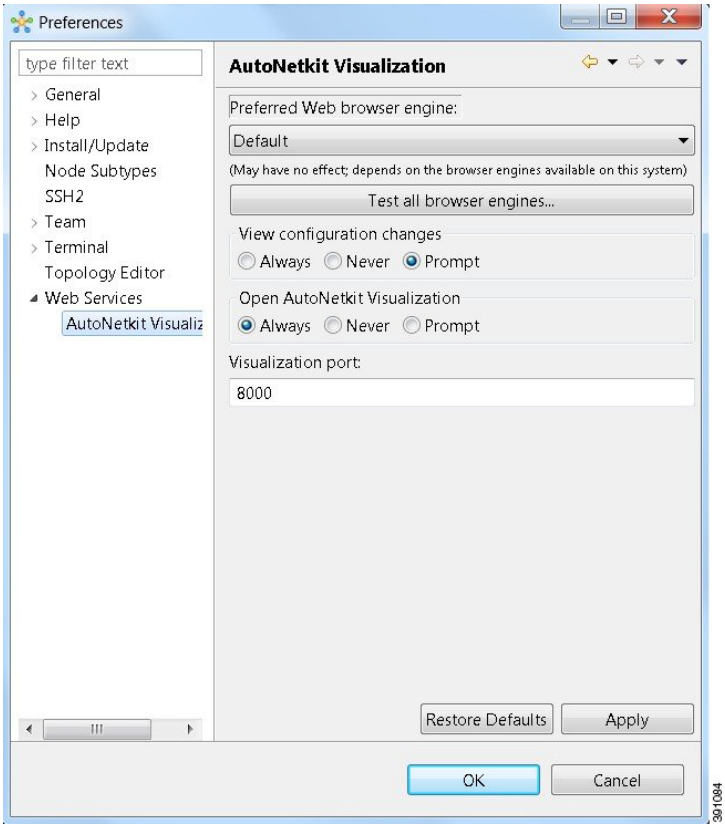
The AutoNetkit Visualization feature is available only when node configurations are built using the parameters defined in AutoNetkit. If AutoNetkit Visualization is turned off, you cannot get a true representation of your topology.

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 42: AutoNetkit Visualization Setting



The available operations for this setting are:

Table 37: AutoNetkit Visualization Setting Operations

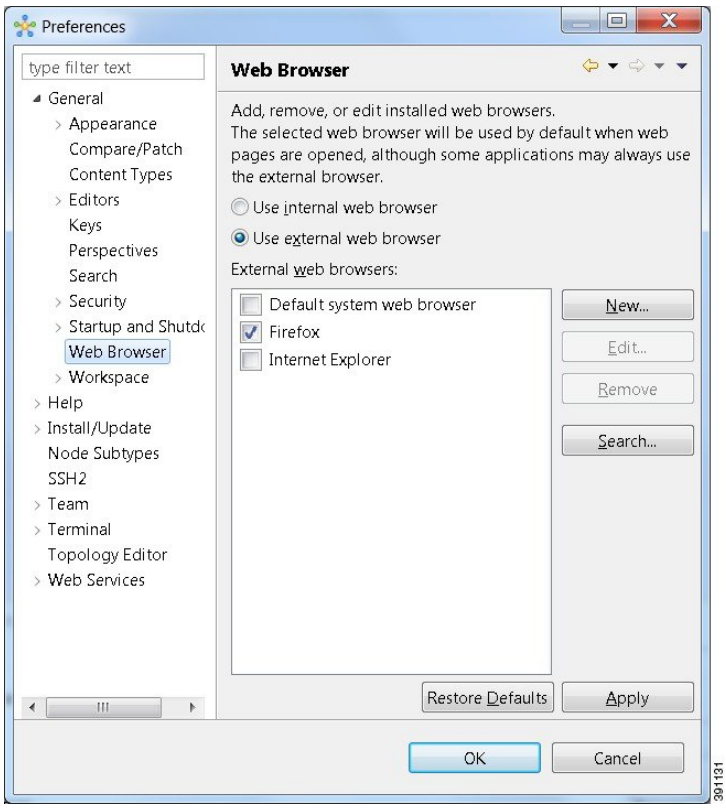
Operation	Description
Preferred Web browser engine	Specifies the external web browser to use when displaying AutoNetkit Visualization results. Options are: • **Default—**the default system browser • **WebKit—**based on the Safari web browser • **Mozilla Firefox** **Note** You are required to use Mozilla 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 43: Web Browser Setting



The available operations for this setting are:

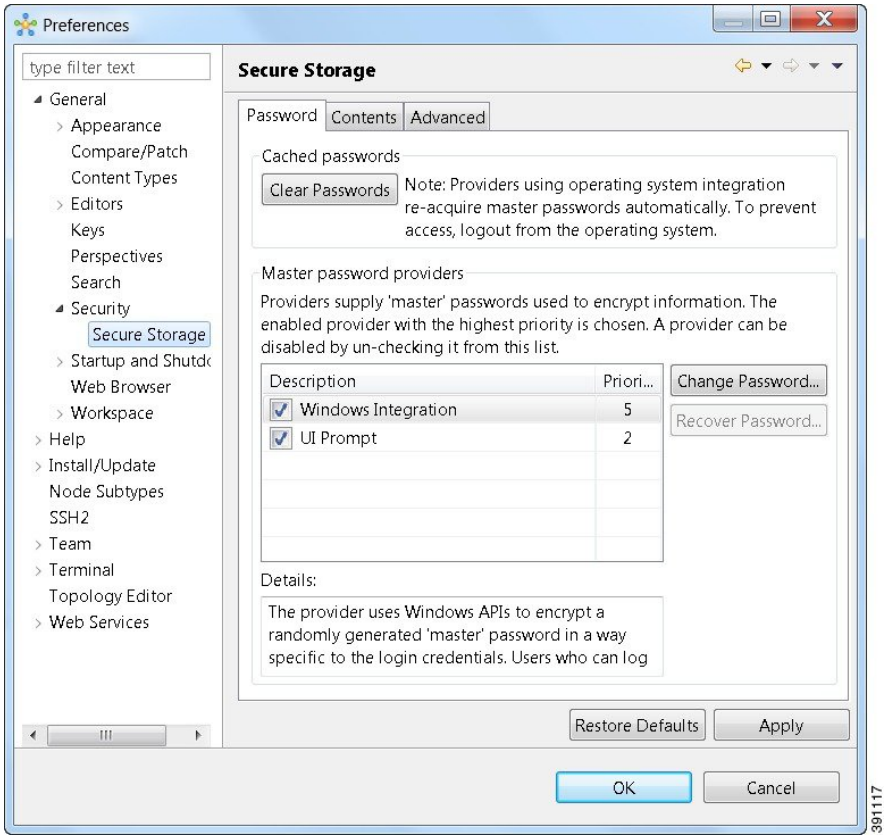
Table 38: Web Browser Setting Operations

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 44: Secure Storage Setting



The **Password** tab pools functionality related to the master password life-cycle and password providers. The available options are:

Table 39: 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.

CHAPTER 3

Design a Topology

Design a Topology Overview

The design phase is the initial step in creating a network topology. During the design phase, you will perform the tasks described in the following sections.

Topology Nodes and Connections

The topology you design consists of nodes and connection functions. Refer to the section Navigating Within the Cisco Modeling Labs Client, on page 8 for additional information about how to select and edit nodes and connection functions.

Topology Nodes

Cisco Modeling Labs provides the Cisco IOSv node subtype. Additional node types are available and can be installed by the system administrator via the User Workspace Management interface. See the User Workspace Management chapter in the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide*.

Table 40: Node Subtypes

Node Name	Node Type
Cisco IOSv	Router node. Runs a Cisco IOS operating system.
Server	Server node. Runs a Linux operating system.
Cisco IOS XRv	Router node. Runs a Cisco IOS XR operating system. (Available separately.)
Cisco CSR1000v	Router node. Runs a Cisco CSR 1000 operating system. (Available separately.)

A node subtype is a virtual machine that runs on top of OpenStack, which itself is running in a Linux virtual machine that is running on top of VMware software. Because the node is virtual, specific hardware is not emulated. For example, there are no power supplies, no fans, no ASICs, and no physical interfaces. For all router nodes, the interface type is a Gigabit Ethernet network interface. A server node has an Ethernet network interface.

You can choose an image and image flavor for each node type. Refer to the User Workspace Management chapter in the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide* for information on how to access the VM Image and the VM Flavor choices. In most cases, you need not select an image and flavor. By default, the node subtype is associated with an image and flavor that runs with the topology.

Table 41: Node VM Images

VM Image Name	Used For
server	Server node
CSR1000v	Cisco CSR1000v node
IOSv	Cisco IOS node
IOS XRv	Cisco IOS XR node

Table 42: Node VM Flavors

VM Flavor Name	Used For
m1_tiny	Linux server
m1_small	Linux server
m1_medium	Linux server
m1_large	Linux server
m1_xlarge	Linux server
server	Linux server
CSR1000v	Cisco CSR 1000v node

VM Flavor Name	Used For
IOS XRv	Cisco IOS XR node
IOSv	Cisco IOS node

Each Linux flavor provides a different amount of memory and CPU allocated to the server.

Connection Functions

Cisco Modeling Labs provides the connection functions shown in the following table.

Table 43: Connection Functions

Connection Type	Description
Connection	Creates a connection between two interfaces. Interfaces are created in the node to support a connection. Any unused interfaces present are automatically assigned. All the interfaces in router nodes are represented as Gigabit Ethernet interfaces. Multiple parallel connections are supported.
Multipoint Connection	Creates a multipoint connection point. Multiple nodes can connect to a multipoint connection.
Layer 3 External (SNAT)	Creates a Layer 3 external connection point using static network address translation (SNAT). This external connection point allows connections outside of Cisco Modeling Labs to connect to the topology.
Layer 2 External (Flat)	Creates a Layer 2 external connection point using FLAT. This external connection point allows connections outside of Cisco Modeling Labs to connect to the topology.

Table 44: Site Groups

Type	Description
Site	Creates a site that can contain one or more nodes, and additional groups of nodes. A site is a group of nodes and you can create multiple site layers.

Create a Topology Project

Cisco Modeling Labs client provides a project folder and a sample topology that you can work with in addition to creating your own project and topology.

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.

The three methods available for creating a topology project are discussed in the following sections.

Before You Begin

Before you begin the task of creating a topology project, you must complete the installation steps for your system and set up the web services profile for the Cisco Modeling Labs client. Refer to the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide*. See Navigating Within the Cisco Modeling Labs Client, on page 8 for information on how to navigate the perspectives and views, and information about the menu bars, toolbars, and editors. The navigation information is also available in the online help.

In addition, understand how:

- To navigate Cisco Modeling Labs client.

Method 1: Create a Topology Project from the Menu Bar

Step 1 From the menu bar, choose **File** > **New** > **Other** > **Topology Project**.

Step 2 Enter a Project Name and select the location in which to save the folder. A project folder is created in the default location if you check the **use default location** check box. Alternatively, you can choose to save the project folder in another location.

Step 3 Click **Finish**.

Method 2: Create a Topology Project from the Projects View

Step 1 Right-click **Projects** view.

Step 2 Choose **New** > **Topology Project**.

Step 3 Enter a Project Name and select the location in which to save the folder. A project folder is created in the default location if you check the **use default location** check box. Alternatively, you can choose to save the project folder in another location.

Step 4 Click **Finish**.

Method 3: Create a Topology Project from the Toolbar

Step 1 Click the **New Topology Project** icon in the toolbar.

Step 2 Enter a Project Name and select the location in which to save the folder. A project folder is created in the default location if you check the **use default location** check box. Alternatively, you can choose to save the project folder in another location.

Step 3 Click **Finish**.

What to Do Next

Create a new topology file.

Create a Topology

Before You Begin

A topology project folder must exist.

There are several methods for creating a topology. These are discussed in the following sections.

Method 1: Create a Topology from the Menu Bar

Step 1 From the menu bar, choose **File** > **New** > **Other** > **Topology**.

Step 2 Select a topology project folder.

Step 3 Enter a filename, ensuring that it ends with the extension **.virl**.

Step 4 Click **Finish**.
A *filename*.virl topology file is created in the selected project folder.

Method 2: Create a Topology from the Projects View

Step 1 Right-click **Projects** view.

Step 2 Choose **New** > **Topology**.

Step 3 Select a topology project folder.

Step 4 Enter a filename, ensuring that it ends with the extension **.virl**.

Step 5 Click **Finish**.
A *filename*.virl topology file is created in the selected project folder.

Method 3: Create a Topology from the Toolbar

Step 1 Click the **New Topology File** icon in the toolbar.

Step 2 Choose **New** > **Topology**.

Step 3 Select a topology project folder.

Step 4 Enter a filename, ensuring that it ends with the extension **.virl**.

Step 5 Click **Finish**.
A *filename.virl* topology file is created in the selected project folder.
Note *filename.virl* is a placeholder for example purposes only.

What to Do Next

Place the nodes.

Place the Nodes on the Canvas

Before You Begin

- A topology file must exist.
- The topology file must be open and the canvas visible in the Topology Editor.

Step 1 Click a node type, which is under the Nodes heading in the **Palette** view.

Step 2 Click the canvas at each point where you want a node. You can also drag the nodes on the canvas to position them. You can then arrange the nodes using several methods:

- Use Shift-click to select two or more nodes. Alternatively, click and drag a selection box around two or more nodes. You can also use Ctrl+click (Windows) or Cmd+click (Mac OS X) to toggle between node selections.
- Use the **Distribute Nodes** tool on the toolbar to arrange the selected nodes vertically, horizontally, or to distribute them on a grid.

  Note The grid is determined dynamically, based on the selected nodes.
- Use the **Align Nodes** tool on the toolbar to arrange the selected nodes in a variety of alignments that you choose from the **Alignment** drop-down list.

  Note The use of **Distribute Nodes** and **Align Nodes** is mutually exclusive.
- When no nodes are selected, use the **Layout Nodes** tool on the toolbar to arrange the nodes in a Tree Layout or a Fruchterman-Reingold (F-R) Layout.
  - A basic tree layout is to choose a node as the root of the tree, and the nodes connected to the root become children of the root node.
  - A F-R layout has connected nodes close to each other, while unconnected nodes are further apart from each other.

See the section Toolbar, on page 13 for information on using these tools.

What to Do Next

Create connections and interfaces.

Create Connections and Interfaces

Before You Begin

Nodes must be in place on the canvas of the Topology Editor.

Step 1 Click **Connection** in the **Tools** view.

Step 2 Click the first node.

Step 3 Click the next node to create the connection.

Note When you connect two nodes, the interfaces are created and named automatically when you choose **File** > **Preferences** > **Topology Editor** > **Associate new connection with interfaces silently** check box. You can view the node interfaces when you click the connection and the two end points of the connection appear in the **Node Editor**.

Step 4 Repeat Step 2 and Step 3 until all the connections are in place.

Tip You can create multiple parallel connections between two nodes. When a connection is selected, the **Topology Editor** shows the connection end points and the **Node Editor** shows the two nodes involved in the connection, all of the interfaces on those nodes, and all the connections between those two nodes.

Note Choose **Export** > **Export Topology Diagram to Image** to capture an image of the current topology on the canvas.

What to Do Next

Create multipoint connections (optional).

Create Multipoint Connections

Multipoint connections are used in BUS topologies where all the nodes are connected to a single transmission medium. This acts as the backbone of the connection and links all the nodes in the network. For example, for four nodes to be directly connected to one another, 12 separate interface connections are needed, three in each router, to fully connect each router to the other. By using a multipoint connection, each router has one connection to the multipoint node.

Before You Begin

A topology file with the extension .virl must exist. Router nodes or server nodes are placed on the canvas. Optionally, connections may exist between nodes.

Step 1 In the **General** view, click **Multipoint Connection**.

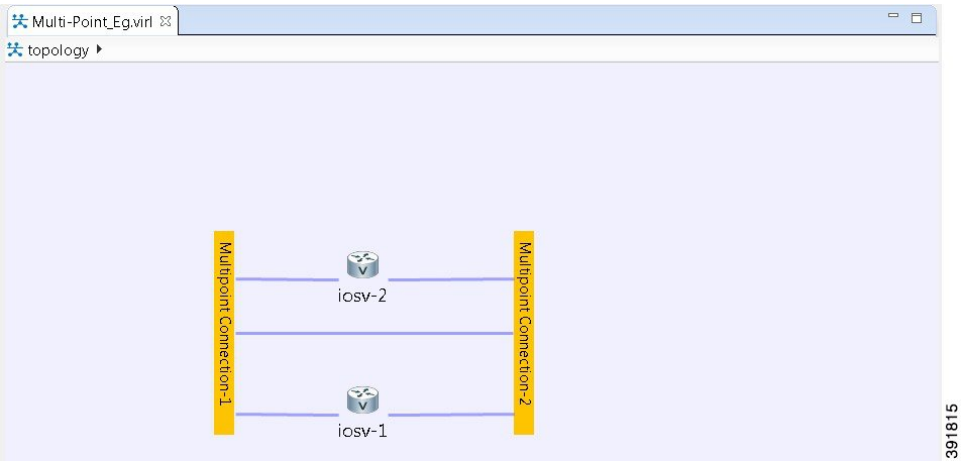
Step 2 Click the area on the canvas where you want the multipoint connection node to appear.

Step 3 In the **Tool** view, click **Connect**.

Step 4 On the canvas, click the multipoint connection node then click an end node. A connection appears. Continue clicking multipoint-node combinations until all connections are made.

Note You can have a connection between two multipoint connection nodes as shown in the following figure.

Figure 45: Creating a Multipoint Connection



Note When performing alignment operations for multipoint connections on the canvas of the **Topology Editor**, occasionally, these do not work as expected.

What to Do Next

- Create sites (optional).
- Show map background (optional).

Create a Site

A site is a container that can hold one or more nodes. It is used to group multiple nodes within the same site, which provides a degree of hierarchy and simplifies the topology view.

A node interface within a site can connect to another node interface in the same site or to a node interface located in a different site. A connection cannot start on a node and terminate on a site. Sites can be nested, forming a hierarchical structure.

Before You Begin

One or more nodes should be on the canvas.

Step 1 In the **General** view, click **Site**.

Step 2 Click an area on the canvas to place the site.

Step 3 **Shift**-click the nodes that you want in the site, or click and drag a selection window around the nodes.

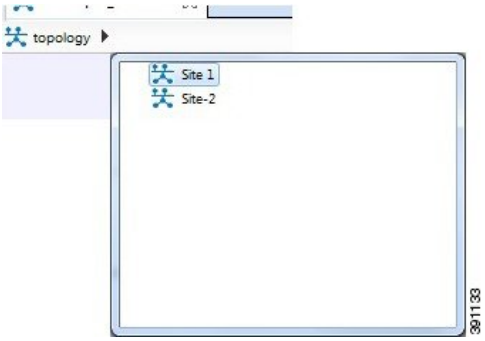
- From the **Edit** menu, choose **Group to Site**.
- Alternatively, you can drag the selected nodes to the site icon that you placed on the canvas.

Step 4 To view the site, select the site name from the drop-down list under the Topology tab.
A topology site tab opens and displays the site view.

The drop-down list under the Topology tab displays the site hierarchy as a navigation trail.

To open a site view, double-click on the site. The following figure shows the navigation trail selections for Site 1 and Site 2.

Figure 46: Site Selection



Step 5 To ungroup the site, click the site icon and choose **Edit** > **Ungroup Site**.

Step 6 To connect node interfaces between Site 1 and Site 2:

a) Select **Site 1**.
b) Click **Connect**.
c) Click the node that is the starting point of the connection, Site 1.
d) Move the cursor to the breadcrumbs and click **Site 2**.
Site 2 is displayed.
e) Click the node that is the endpoint of the connection.
A connection now exists between the interfaces in the two sites.

Step 7 To delete a node from a site:

a) Select the applicable node.
b) Right-click the node and select **Delete** from the list.
The node is deleted.

Step 8 To delete a site from a topology:

a) Select the applicable site.
b) Right-click the site and select **Delete** from the list.
The site is deleted including all associated interfaces and connections.

What to Do Next

Continue designing the topology.

View the Topology on a Map

The topology view is a schematic diagram. It shows the nodes and connections arranged in a logical fashion, or represents a design that is easy to view. When you create a map view, you can move the nodes such that they represent a physical location, for example, the locations of all router nodes in a city. Moving nodes on a map view does not change their location in the topology view.

Within a topology file, each node is represented by two sets of coordinates. The location coordinates define the node placement in the topology view. The map coordinates define node placement on the map view, by latitude and longitude.

The map view uses the MapQuest database for map display.

Before You Begin

- A topology must exist with nodes and connections in place.

- You must be connected to the Internet to use the mapping function, since it uses the map service from MapQuest to retrieve the latest map images.

Step 1 With the topology view active, select **Edit** > **Show map Background**. Alternatively, you can click the **Show Map Background** tool in the toolbar.

Step 2 Before you move a node, select the desired map location.

a) In the **Map** view, click the **Go To Location** icon, located under the pan and zoom controls.

- Type in the first few characters of a location.
- On a Windows platform, press Ctrl-Space and a list of matching locations appears.
- You can enter an airport code.

Note We recommend that you enter all upper-case or initial cap words when searching for locations, as these provide more reliable results. For example, enter MEL or Mel to search for Melbourne airport.

b) Select and click a location from the list.

c) Click **OK**.
The map location is displayed.

d) Use the pan and zoom controls on the map to determine an exact location. You can double-click the coordinates in the Status bar, which opens the **Go To Location** dialog box.

Step 3 Click and drag the nodes into position on the map.
When you click and drag a node, the map coordinates are added to the .virl file and the node becomes a part of the map view. When you open the map view again, the map location with the nodes is displayed.

Step 4 Continue to click and drag the nodes into position on the map view.

Step 5 Click the **Show Map Background** tool in the toolbar to revert to the topology view. Note that the nodes in the topology view are in their original layout.

CHAPTER 4

Build a Configuration

Build a Configuration Overview

In the build phase, you build the configurations for each node. After selecting the options for the overall topology and each node, you create the configuration files. Alternatively, you can use AutoNetkit to create the configuration files.

You can modify and save configuration files for the topology and for each node in your topology.

Create and Modify a Node Configuration

While AutoNetkit is useful for generating configuration files for all the nodes in the topology, you can bypass AutoNetkit and enter node configuration information directly.

You can enter configuration information in either of the following ways:

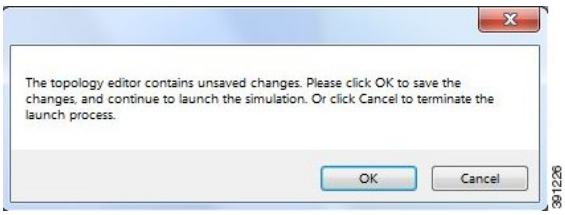
- During the design phase, copy and paste configuration commands for each node.
- During the simulation phase, connect to a node console and change its configuration when the topology is running. See the chapter Simulate the Topology Overview for more information on how to modify, extract, and save a running configuration.

Note When you create your configuration files:

- Changes that are manually entered are not visible in the topology design. If you create a new interface by entering configuration commands, the interface is not created in OpenStack nor does the interface show up in any of the node views.
- Depending on how the AutoNetkit Auto-generate feature is set, you may overwrite the changes you enter.

While in the **Design** perspective, any changes you manually make to a node configuration are saved in the current *filename*.virl file. Before you launch a simulation from the **Design** perspective, a notification window advises you to save the changes or cancel the simulation launch.

Figure 47: Save Changes Before Launch



Create a Node Configuration Manually

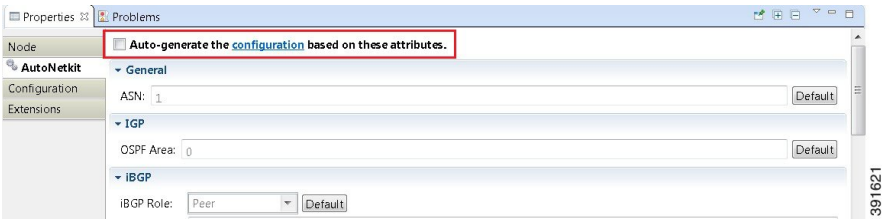
Before You Begin

The topology design should be complete.

Step 1 In the **Topology Editor**, click a node.

Step 2 In the **Properties** view, click **AutoNetkit** and uncheck the **Auto-generate the configuration based on these attributes** check box.

Figure 48: Uncheck Auto-generate Check Box



Step 3 Click **Configuration**.

Step 4 Enter the configuration commands in the **Configuration** view.

Note All changes are automatically saved to the *filename*.virl file. However, the changes made do not appear in the topology on the canvas.

What to Do Next

Launch a simulation to observe the changes.

Use an Existing Node Configuration

You can use an existing configuration file to create a node configuration in Cisco Modeling Labs.

Before You Begin

The topology design should be complete.

Step 1 In the **Topology Editor**, click a node.

Step 2 In the **Properties** view, click **AutoNetkit** and uncheck the **Auto-generate the configuration based on these attributes** check box.

Step 3 Click **Configuration**.

a) Open the configuration file you want to use and copy the configuration commands.

b) In the **Configuration** view, paste the configuration commands.

Note All changes are automatically saved to the *filename*.virl file. However, the changes made do not appear in the topology on the canvas.

What to Do Next

Launch a simulation to observe the changes.

Import the Configuration from a Cariden MATE File

You can import a topology from an existing Cariden MATE file, version 5.2.0 or later. Cisco Modeling Labs client will accept site imports up to two layers deep. Any Cariden MATE file that has a topology with more than two layers of sites will not import correctly.

Before You Begin

A valid Cariden MATE file is available on your file system.

Cisco Modeling Labs client is running.

Your license allows Cariden MATE file import.

Step 1 Choose **File > Import**

A window appears, prompting you to Import topology from the Cariden MATE file.

Step 2 Choose **Import Cariden MATE File** then click **Next**.
The license is checked. When confirmed, click **Next**.

Step 3 Choose the **From data file**Cariden MATE file to import. Use **Browse** to select the directory and file to import.

Step 4 Choose the location **Into target folder** for the Cariden MATE file. Use **Browse** to select the target Project folder.

Step 5 Enter a filename for the imported Cariden MATE file.

Important The filename you enter must have the extension **.virl**. For example, **Lab_import.virl** is a valid filename. Otherwise, you cannot open the file in the topology editor.

The Cariden MATE file converts to a Cisco Modeling Labs **.virl** file.

Step 6 In the Projects view, expand the project folder where you saved the imported file.

Step 7 Right click on the imported file, for example **Lab_import.virl** and choose **Open With > Topology Editor**.
A topology canvas opens and displays the topology.

What to Do Next

Visualize the configuration.

Run the simulation.

Create Node and Interface Configurations Using AutoNetkit

Before You Begin

The topology design should be complete.

Step 1 Verify the configuration for each node in the topology.

a) In the **Topology Editor**, click a node.

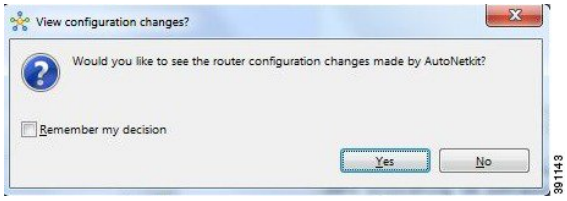
b) In the **Properties** view, click **AutoNetkit**. Verify **Auto-Generate** is checked or unchecked, depending on whether AutoNetkit will generate a configuration for that node.

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

Step 2 Generate a configuration for the topology. Click **Update Router Configurations** from the toolbar. Alternatively, from the menu bar, choose **Run > Update Router Configurations**. You are prompted to save any changes made since the previous configuration update.
If the **Auto-generate** check box is checked for a node, the configuration updates are generated by AutoNetkit.

Step 3 AutoNetkit displays a notification after it generates the configuration. Click **No** to skip a comparison of configuration changes. Click **Yes** to open a comparison view of the configuration changes.

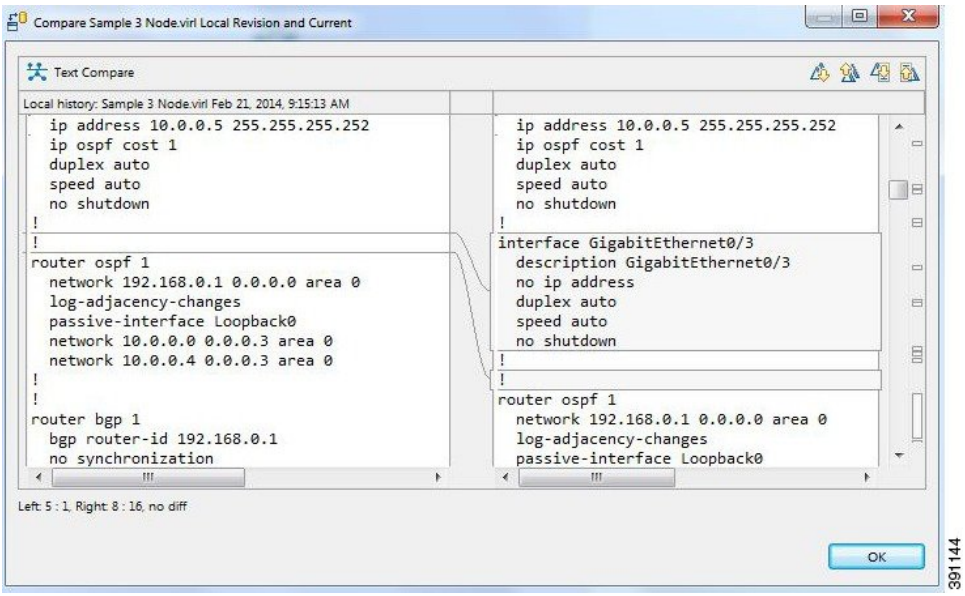
Figure 49: View Configuration Changes? Notification



Note Selecting the **Remember my decision** check box will always display configuration changes for subsequent invocations. You can later change this behavior by choosing **File** > **Preferences** > **Web Services** > **AutoNetkit**.

The .virl file opens and displays previous and current configurations side-by-side, with the changes highlighted. You can scroll through the contents and see the differences. However, you cannot edit the configurations.

Figure 50: Show Configuration Comparison Side-by-Side

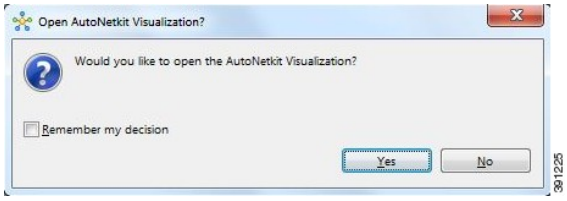


Click **OK** to close the comparison view.

Step 4 When you close the comparison view, a notification is displayed, and you can choose whether or not to open AutoNetkit visualization.

- Click **No** to skip the visualization. You return to the **Design** perspective.
- Click **Yes** to display the visualization. The AutoNetkit visualization opens in a browser window. For more information about this feature, see Visualization Overview, on page 87.

Figure 51: Open AutoNetkit Visualization? Notification



Note Selecting the **Remember my decision** check box will always open AutoNetkit visualization for subsequent invocations. You can later change this behavior by choosing **File** > **Preferences** > **Web Services** > **AutoNetkit**.

CHAPTER 5

Visualizing the Topology

Visualization Overview

The visualization phase allows you to see how the nodes interact in terms of routing protocol connectivity, autonomous system (AS) numbers, Open Shortest Path First (OSPF) area, and so on. Before entering the visualization phase, you must have designed the topology and built the node configurations.

Note Visualization is only available where node configurations are generated using parameters defined in AutoNetkit.

The AutoNetkit visualization runs in a browser window, either within the Cisco Modeling Labs client or in a separate browser window. Ensure that you use a compatible browser, as described in the *Cisco Modeling Labs Installation Guides* for the version of Cisco Modeling Labs that you are using.

The following figure shows an overview of the visualization phase as it appears in a browser window.

Figure 52: Visualization Overview

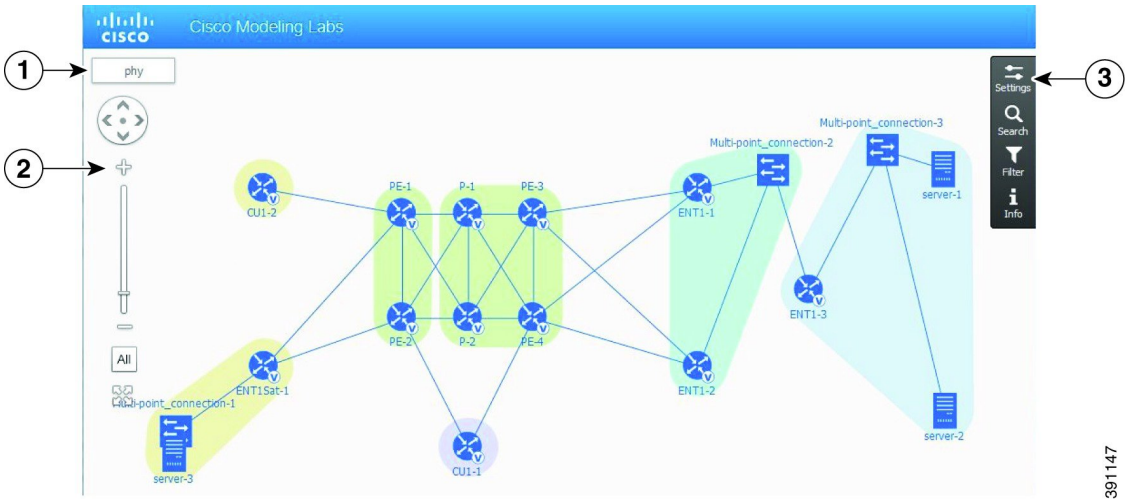
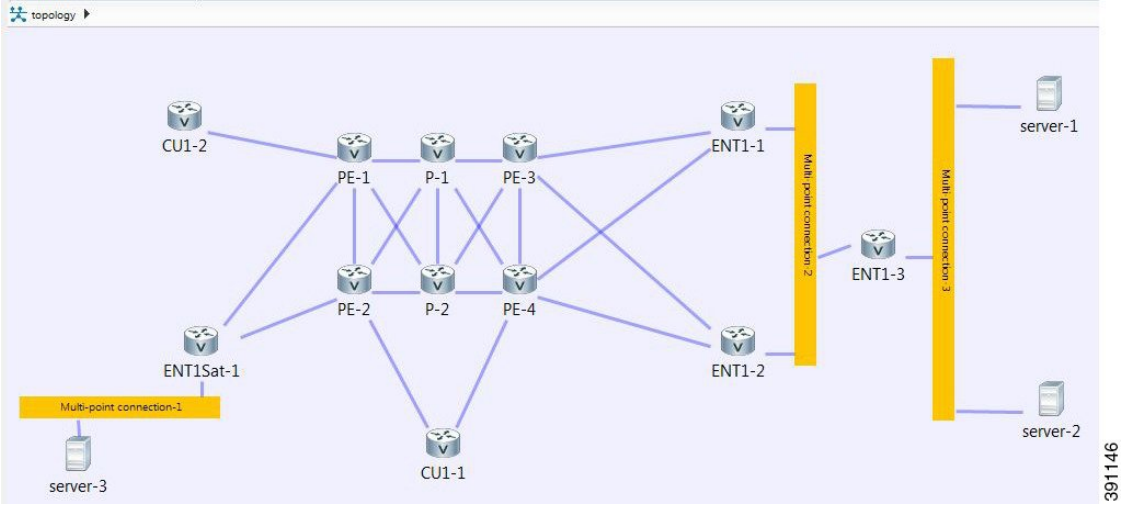


Table 45: Visualization Overview

Identifier	Description
1	Layers view selection. The layers shown correspond to the values selected when generating the configuration. For example, if IGP is chosen as OSPF, an OSPF layer will be shown. If MPLS is not used in the configuration, no MPLS layer will be shown.
2	Pan and zoom controls. Use the controls to pan the topology, zoom in and out on the topology, adjust the display so that all the nodes are shown, and switch to a full-screen view.
3	Settings, search, and filter controls. Use the settings control to change the appearance of the display. Use the search and filter controls to highlight specific nodes and connections.

The following figure shows how the visualization compares to the topology design.

Figure 53: Topology Design



During the visualization phase, you perform the tasks described in the following sections.

Enabling AutoNetkit Visualization (for Windows Users)

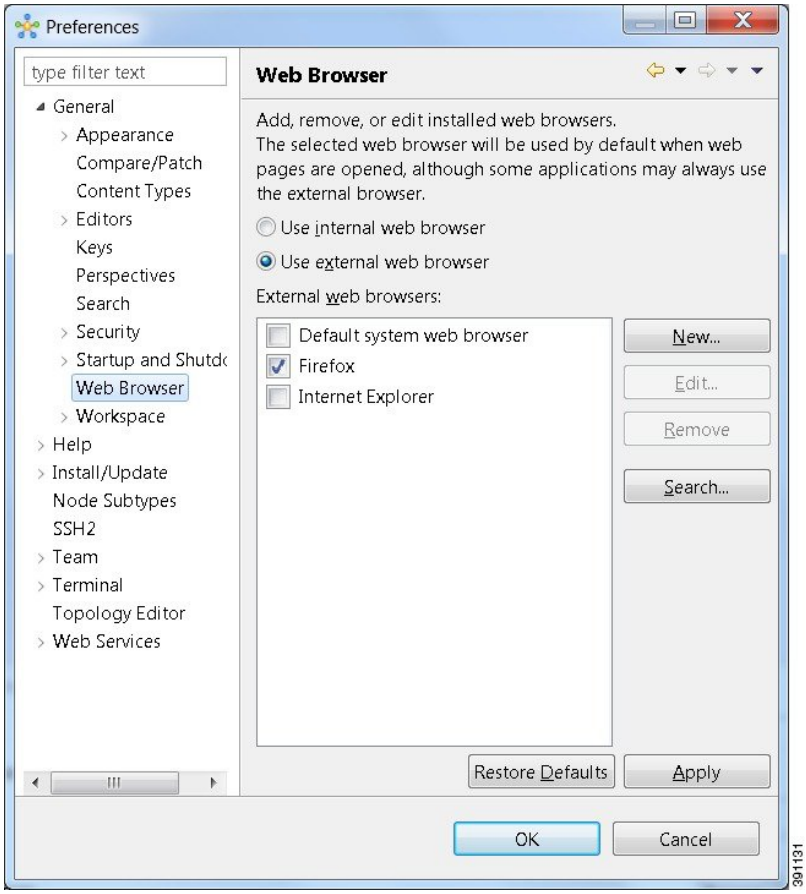
To enable visualization for the Cisco Modeling Labs client on Windows, complete the tasks described in this section.

Before You Begin

- Ensure that you have access to the Cisco Modeling Labs client.

Step 1 From the Cisco Modeling Labs client toolbar, choose **File** > **Preferences** > **General** > **Web Browser**.
The Web Browser Preferences dialog box appears.

Figure 54: Web Browser Preferences



Step 2 Click the **Use external web browser** radio button.

Note On Windows, only Mozilla Firefox, Google Chrome, or Apple Safari are supported as default web browsers. Internet Explorer is not supported for AutoNetkit Visualization.

Step 3 In the **External web browsers** pane, check the **Firefox** check box.

Step 4 Click **Apply** and **OK**.

Step 5 Choose **Preferences** > **Web Services** > **AutoNetkit Visualization**.

Step 6 Under **Open AutoNetkit Visualization**, select the **Always** option.

Step 7 Click **Apply** and **OK**.

If the Firefox executable cannot be found, you will need to edit the path to find it (*firefox.exe*).

a) To do this, in the **External web browsers** pane (Step 3), select **Firefox** and click **Edit**.

b) Click **Browse** to navigate to the corresponding location and choose the Firefox executable.

c) Click **OK** to save the changes.

Opening AutoNetkit Visualization

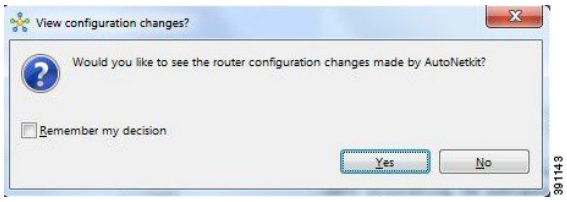
Before You Begin

Complete the task of building nodes and interfaces.

Step 1 Generate a configuration for the topology.
Click **Update Router Configurations** from the toolbar. Alternatively, from the menu bar, choose **Run** > **Update Router Configurations**.

Step 2 View the configuration changes.
AutoNetkit displays a notification after it generates the configuration.

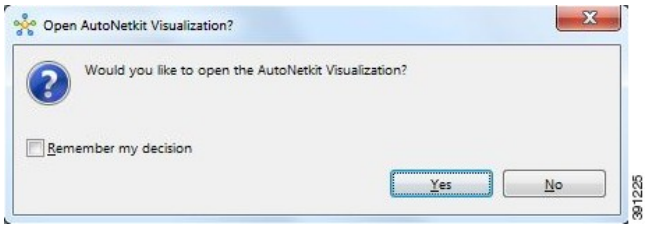
Figure 55: View Configuration Change Notification



- Click **No** to skip this step.
- Click **Yes** to open a comparison view of the configuration changes.

Step 3 Display the AutoNetkit Visualization view of the topology.
When you close the comparison view, a notification prompts you whether or not to open the AutoNetkit Visualization.

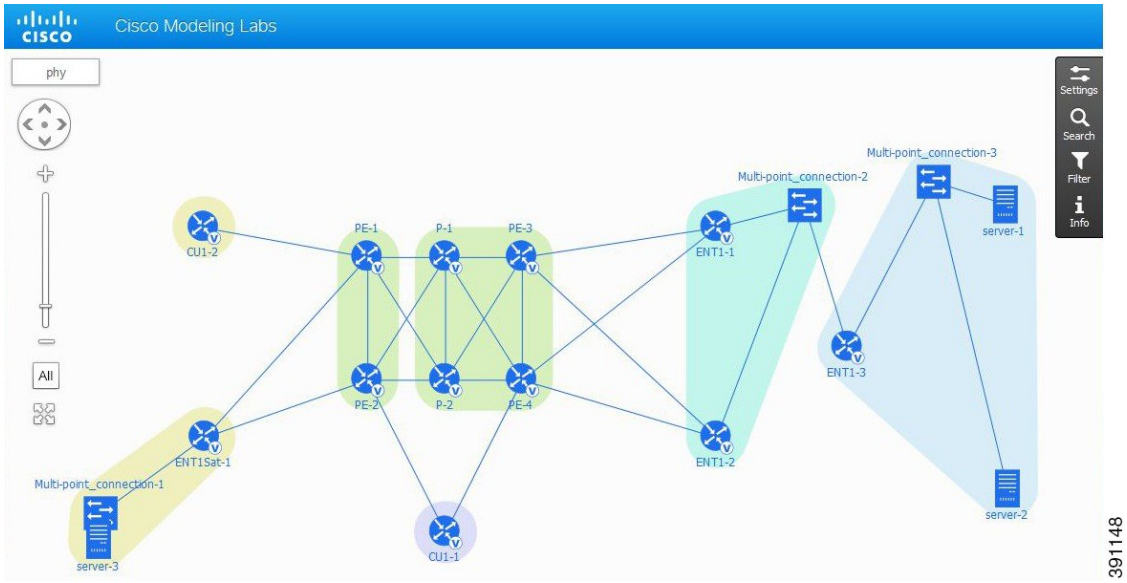
Figure 56: Open AutoNetkit Visualization



- Click **No** to skip this step.
- Click **Yes** to display the visualization.
  The AutoNetkit Visualization opens in a browser window.

Note Choose **File** > **Preferences** > **Web Services** > **AutoNetkit Visualization** to control the prompts for visualization.

Figure 57: AutoNetkit Visualization Window



What to Do Next

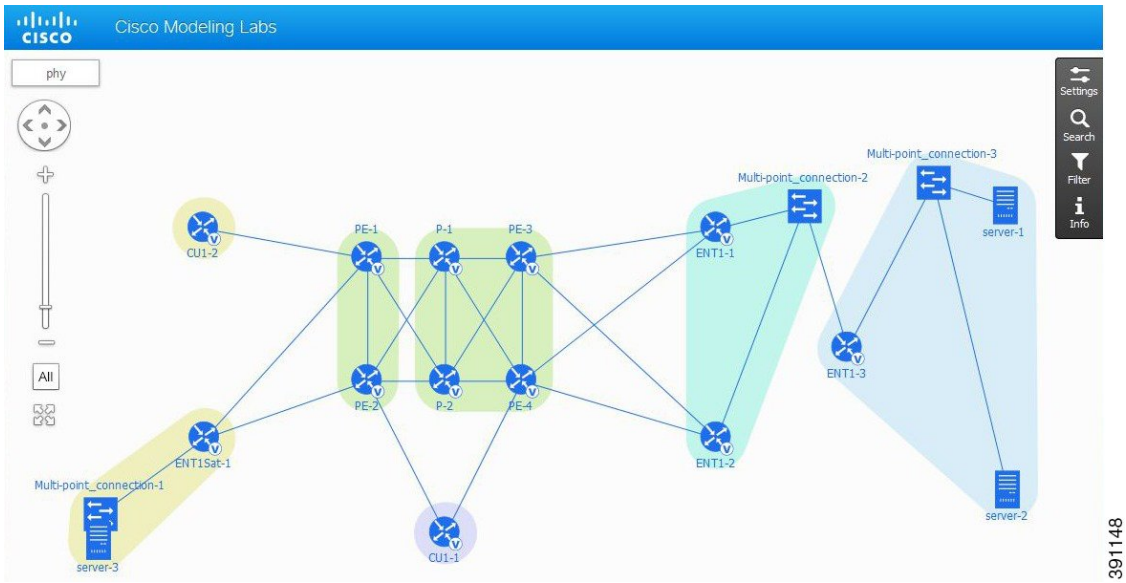
Explore the features in AutoNetkit Visualization.

Using Layers

Step 1 Verify if visualization is open in a browser window.

The initial layer that is displayed in the browser window is the physical model of the topology, as shown in the following figure. The physical model shows the nodes and interface connections between the nodes. It is similar to the Cisco Modeling Labs topology view.

Figure 58: Physical Layer View

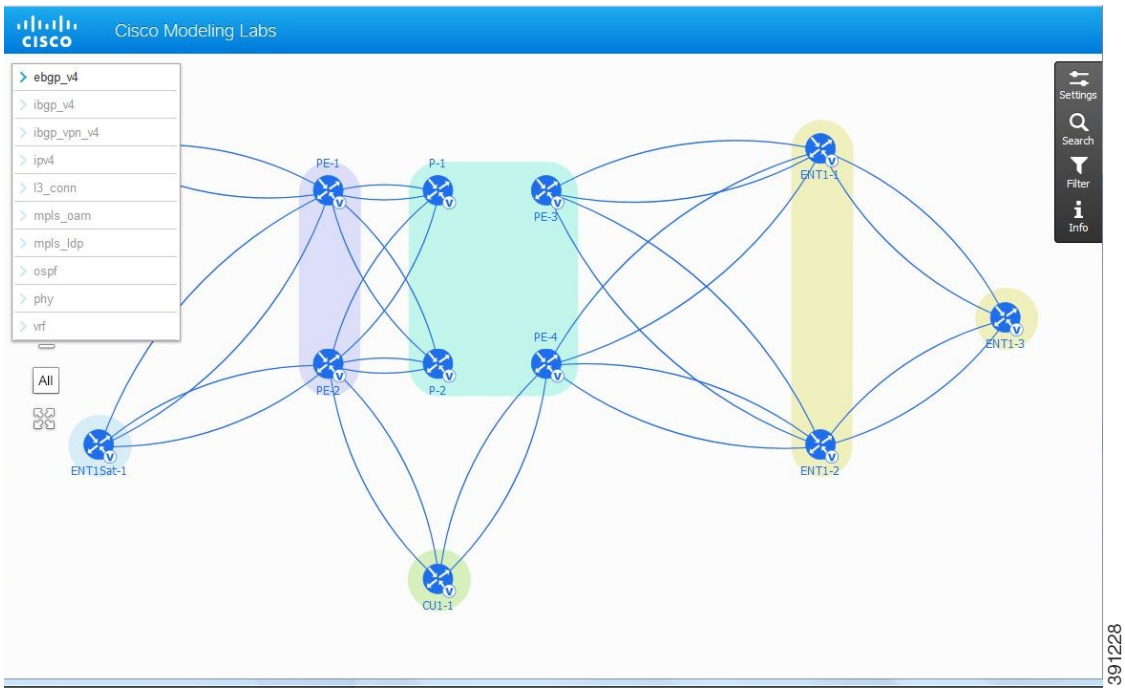


Note The options change depending on the router protocols configured. For example, if IPv6 is configured, you will also see **ebgp_v6**

Step 2 To select another view, place the cursor over the **Layers** view selection in the browser window. See Visualization Overview, on page 87 for information on the **Layers** view selection.

When you place the cursor over the layers view, several choices appear. For example, selecting **ebgp_v4** will show the IPv4 eBGP topology. This is constructed based on the AS property and node connections created in the **Topology Editor**, as shown in the following figure.

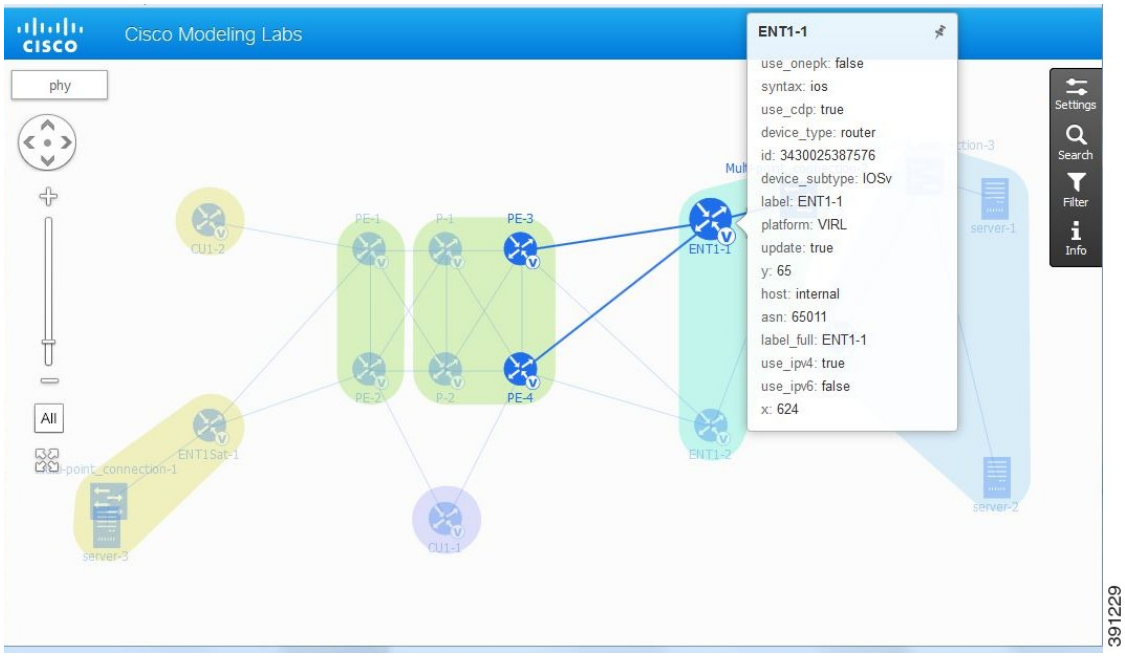
Figure 59: ebgp_v4 Layers View



Step 3 Place the cursor over one of the nodes.
This action displays a pop-up view of information about that node. The type of information displayed depends on the selected layer and node configuration.

Note You can also hover over the connections to see connection details.

Figure 60: Node Pop-Up Information



Step 4 Continue selecting layer views and observe how the protocol-centric view changes. In a complex topology, you can use the **Layers** views to verify that the protocols, nodes, and connections meet the design requirements.

What to Do Next

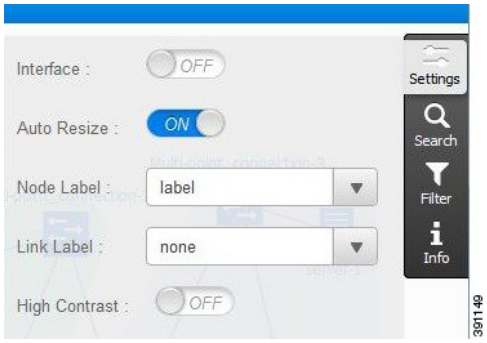
- (Optional) Change settings, use the Search feature, or use the Filter feature.
- (Optional) Return to the design and build phases to modify the topology and generate new configurations and AutoNetkit Visualizations.
- (Optional) Run the topology simulation when you are satisfied with your configuration.

Changing the Settings

Step 1 In the AutoNetkit Visualization browser window, click **Settings**.

The Settings window opens, as shown in the following figure.

Figure 61: AutoNetkit Visualization Settings Window



The following table lists the main settings.

Table 46: Main Settings

Setting	Description
Interface	Select **ON** to display the interface connection points. The default value is **OFF**.
Auto Resize	Select **OFF** to not resize the visualization automatically. The default value is **ON**.
Node Label	Select a value from the drop-down list. The default value is **label**, which is the node name. This node name is configured in Cisco Modeling Labs for each node.
Link Label	Select a value from the drop-down list. The default value is **none**.
High Contrast	Select **ON** to change the visualization display to a high-contract color scheme. The default value is **OFF**.

Step 2 Observe the changes in the display when you select different settings.

Step 3 To close the **Settings** view, click the Settings tab.

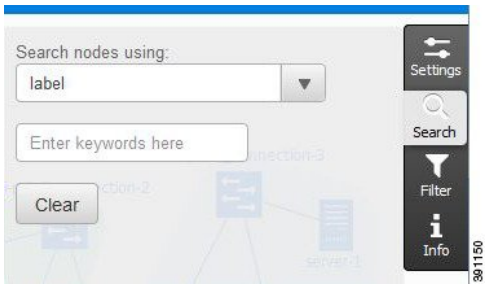
What to Do Next

- (Optional) Change settings, use the Search feature, or use the Filter feature.
- (Optional) Return to the design and build phases to modify the topology and generate new configurations and AutoNetkit Visualizations.
- (Optional) Run the topology simulation.

Using Search

Step 1 In the AutoNetkit Visualization browser window, click **Search**.
The **Search** window opens.

Figure 62: AutoNetkit Visualization Search Window



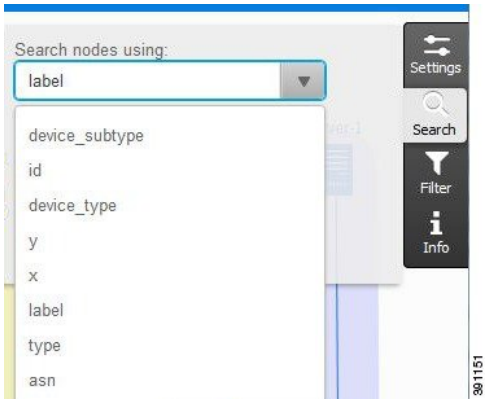
The **Search** window contains the main options that are listed in the following table.

Table 47: Search Window Main Options

Option	Description
Search nodes using	Selects a search attribute from the drop-down list. The default value is **label**.
Enter keywords here	Identifies keywords based on the attribute selected in the **Search nodes using** drop-down list. Keywords are case-sensitive. Wildcard selections are not supported. For example, searching **asn** with the keyword **65*** does not modify the display, even if ASN values of 65000 and 65001 exist in the topology configuration. Partial matches are not supported for numeric node values. Partial matches are supported in alphanumeric fields. For example, selecting **label** and entering the keyword **P** matches all the node labels that begin with a capital P or have a capital P anywhere in the label.
Clear	Clears the keywords.

The following figure shows the values you can select from the **Search nodes using** drop-down list.

Figure 63: Search Window Drop-Down List



Step 2 Observe the visualization and how the different items are highlighted when they match the combination of **Search nodes using** and **keywords** that you enter.

Step 3 To close the **Search** window, click the **Search** tab.

What to Do Next

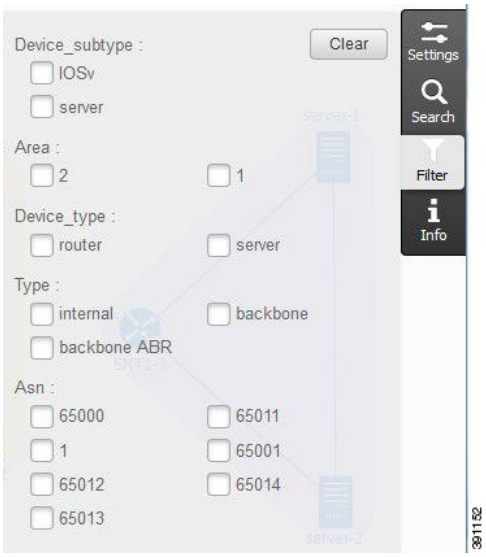
- (Optional) Change settings, use the Search feature, or use the Filter feature.
- (Optional) Return to the design and build phases to modify the topology and generate new configurations and AutoNetkit Visualizations.
- (Optional) Run the topology simulation.

Using Filters

Step 1 In the AutoNetkit Visualization browser window, click **Filter**.

The **Search** window opens.

Figure 64: AutoNetkit Visualization Filter Window



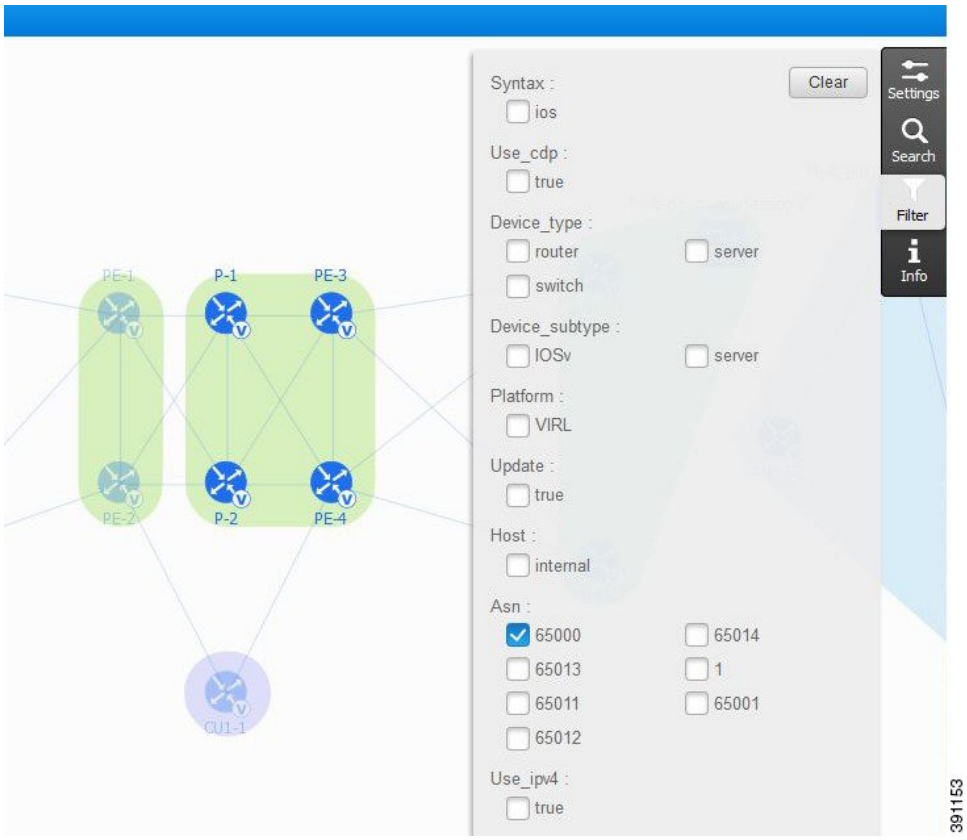
The **Filter** window contains the main options that are listed in the following table.

Table 48: Filter Window Main Options

Option	Description
Clear	Clears all the checked selections.
Selections	The selections that are available for filtering depend on the topology configuration. Values that do not exist in the topology configuration do not appear in the selection window. **Note** You can further drill down or expand your search using the parameters **OR** within a keyword and **AND** between keywords.

For example, if your configuration contains multiple AS values, you can filter the display to show only AS values that match 65000.

Figure 65: Filter AS Value 65000



Step 2 To close the Filter window, click the **Filter** tab.

What to Do Next

- (Optional) Change settings, use the Search feature, or use the Filter feature.
- (Optional) Return to the design and build phases to modify the topology and generate new configurations and AutoNetkit Visualizations.
- (Optional) Run the topology simulation.

CHAPTER 6

Simulate the Topology

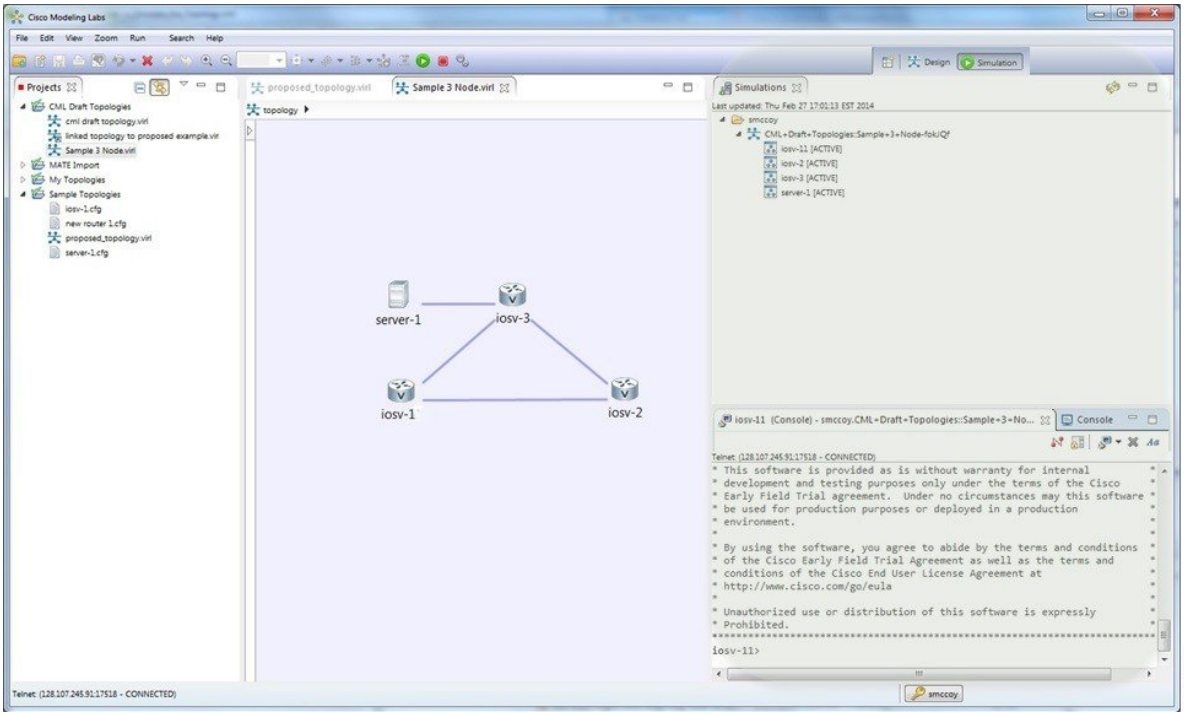
Simulate the Topology Overview

The simulation phase is when you run the simulation of your topology design. The **Simulation** perspective provides a set of views that support the simulation phase. By comparison, the design and build activities occur in the **Design** perspective, which provides a set of views that support the design activity. Some views in the **Simulation** perspective can also be viewed in the **Design** perspective.

Simulation Perspectives and Views

The main areas of focus within the **Simulation** perspective are the **Simulations** view and the **Console** view. The following figure highlights the **Simulation** and **Console** views for a running simulation.

Figure 66: Simulation Perspective Initial Display



391154

From the **View** menu, you can open additional views in the perspective and arrange the open views by dragging them within the perspective workspace. See the online help and the section Customizing Perspectives, on page 19 for more information.

Note To reset your current perspective to its original configuration when the workbench was first opened, right-click the perspective button and select **Reset**.

Node Limits

In Cisco Modeling Labs 1.0 Corporate Edition, the number of nodes you can run is limited by the resources available on the server and the number of licenses available. In Cisco Modeling Labs 1.0 Corporate Edition, the design limit is 1,000 nodes.

During the simulation phase, you can perform the tasks described in the following sections.

Launch a Simulation

Before You Begin

- Complete the topology design.
- Complete the task of building the nodes and interfaces.
- (Optional) Generate the configuration using AutoNetkit.

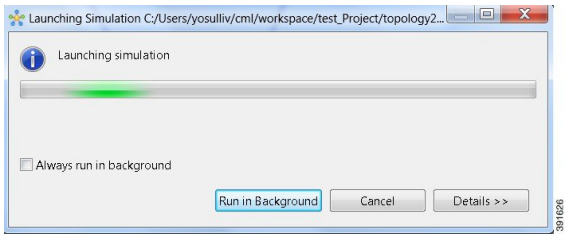
Caution When you manually make changes to a node configuration and bypass AutoNetkit auto-generation, those changes do not appear in the topology view of the **Design** perspective or **Simulation** perspective. For example, if you use the **hostname** command to change the host name from `iosv-1` to `Router-1` in the configuration, the node name in the topology view and in other related views remains as `iosv-1`.

- Open the desired topology.

Note The topology should be open and visible in the **Topology Editor**. If you have multiple topologies open in the **Topology Editor**, simulation will launch for the currently active view.

Step 1 From the toolbar, click the **Launch Simulation** button.
The simulation launches and provides a unique identifier, which means that multiple instances of the topology can be launched and each will have a unique name.

Step 2 In the **Launching Simulation** dialog box, select any of the following actions:



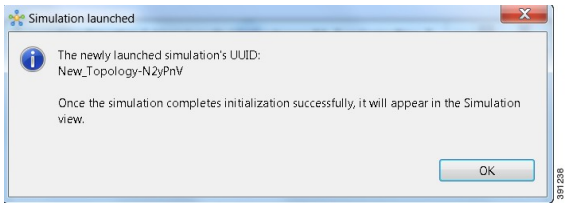
a) Check the **Always run in the background** check box. All future node start requests, stop requests, and simulation launch requests run in the background and do not display dialog boxes.
   Note To control the background setting, choose **File** > **Preferences** > **General**.

b) Click **Run in background**. The dialog box closes when the node simulation starts.

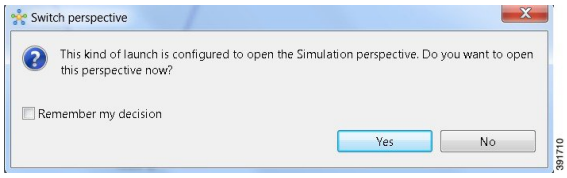
c) Click **Cancel** to return to the **Design** view.

d) Take no action, and the node simulation launches momentarily.
   Tip When you click **Run in background**, the status bar displays a progress icon. Click the icon to display a compact view of the activity progress. If the background activity encounters an error, the icon shows a red **X**. Click the error icon to display the error dialog.

Step 3 In the **Simulation launched** dialog box, click **OK**.



Step 4 When prompted to switch to the **Simulation** perspective, click **Yes**.

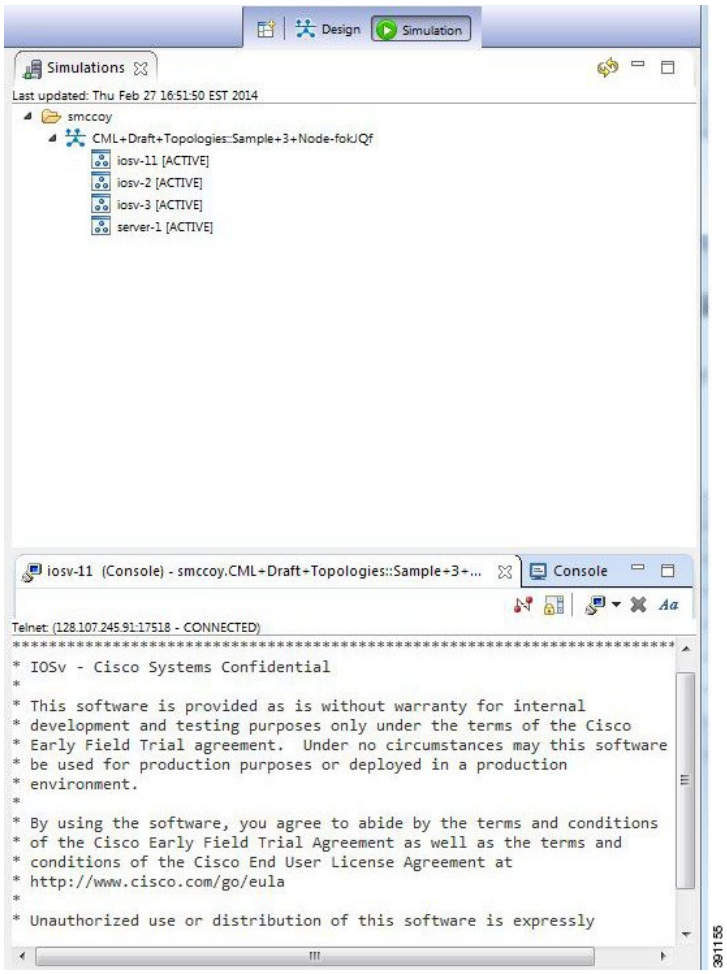


Note We recommend that you switch to the **Simulation** perspective to view the running simulation and to connect to node consoles.

The Cisco Modeling Labs client tracks the state of the simulations. All launched simulations appear in the **Simulations** view. Status messages are displayed in the **Console** view. After the Cisco Modeling Labs server has started the simulation launch, a confirmation dialog box appears with an identifier assigned to the simulation by the Cisco Modeling Labs server.

When the nodes are running on the Cisco Modeling Labs server, they are displayed in the **Simulations** view with the status as [ACTIVE], as shown in the following figure.

Figure 67: Simulation Launched in Simulation View



Connect to a Simulation Node Console

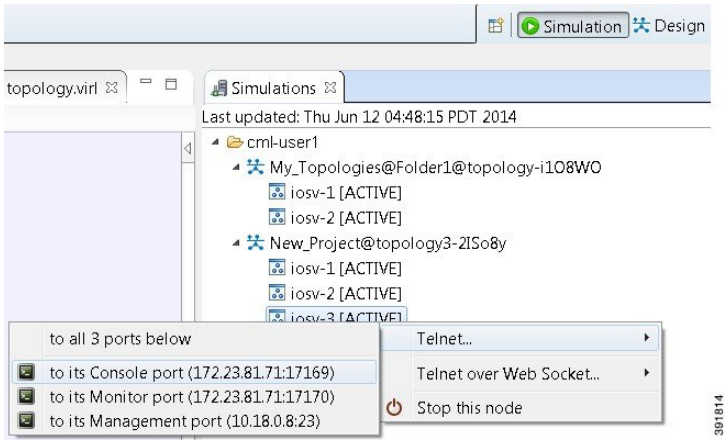
Before You Begin

- Launch a simulation.
- Ensure that the **Simulation** perspective is active.

- Ensure that the **Simulations** view and **Console** view are displayed.

Step 1 To connect to a console for a specific node, right-click the node in the **Simulations** view and choose **Telnet** > **to its Console port**.

Figure 68: Connecting to a Node Console



A new **Terminal** view opens.

Figure 69: Terminal View

Step 2 To disconnect a terminal from the simulation, click **Disconnect** in the **Terminal** view toolbar or click the **Close** icon in the **Terminal** view.

Note When you disconnect or close a terminal view, all text in the view is discarded.

Tip After you disconnect a terminal, but do not close the terminal window, you can press **Enter** to reconnect the terminal.

Stop an Entire Simulation

There are several methods for stopping an entire simulation. These are discussed in the following sections.

Method 1: Stop an Entire Simulation from the Toolbar

Step 1 In the toolbar, click the **Stop Simulations** button.
A **Stop Simulation(s)** dialog box appears.

Step 2 In the **Simulation** column, click once to highlight the simulation to stop.

Step 3 (Optional) To save the configurations, click the adjacent setting in the **Extract Configurations?** column until the prompt changes to **Yes**.

Note Configurations for server nodes are not extracted.

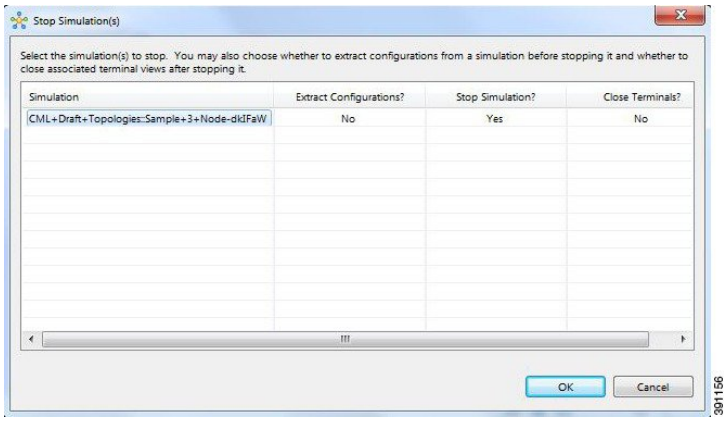
Step 4 (Optional) To close the internal terminals associated with the simulation, click the adjacent setting in the **Close Terminals?** column until the prompt changes to **Yes**.

Note External terminal connections are not stopped as part of this operation and must be closed manually.

Step 5 To stop the simulation, click the adjacent setting in the **Stop Simulation?** column until the prompt changes to **Yes**.

Step 6 Click **OK** to confirm stopping the simulation, or click **Cancel** to leave the simulation running.

Figure 70: Stop Simulation(s) Dialog Box



On Mac OS X, you update the values for **Extract Configurations?**, **Stop Simulation?** and **Close Terminals?** in the columns directly. You do not need to select the name of the simulation.

Note When stopping a simulation, you cannot attempt to stop individual nodes running in the simulation at the same time. This behavior is not supported.

Method 2: Stop an Entire Simulation from the Simulations View

Step 1 In the **Simulations** view, right-click the **Simulation** name and select **Stop Simulation**. The **Stop simulation confirmation** dialog box appears.

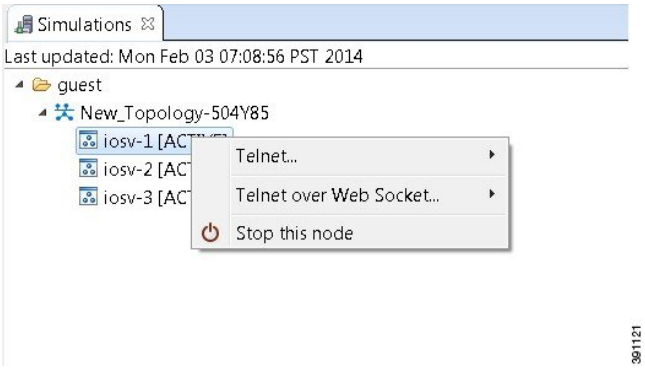
- (Optional) Check the **Extract configurations before stopping simulation** check box to save the current configurations.
- (Optional) Check the **Close associated terminal views** check box to close all the open internal terminals associated with the specific simulation. External terminals are not closed.

Step 2 Click **OK** to stop the simulation.
Once selected, all nodes in the simulation start shutting down. It may take a few minutes for the simulation to shut down completely and to disappear from the **Simulations** view.

Note When stopping a simulation, you cannot attempt to stop individual nodes running in the simulation at the same time. This behavior is not supported. Note too for instances where a user account expires, any running simulations for that user continue to run. Since the user account can no longer log into stop them, they will remain active until the next system reboot or until the system administrator explicitly stops them.

Stop a Single Node

Step 1 In the **Simulations** view, right-click the node to stop and select **Stop this node**.



The **Are you sure** dialog box appears.

Step 2 Click **OK** to stop the simulation. Alternatively, click **Cancel** to abandon the operation and return to the simulation.

Note When you click **OK**, the node stops without saving any changes to the configuration.

When a node is stopped, its status changes to [DELETED].

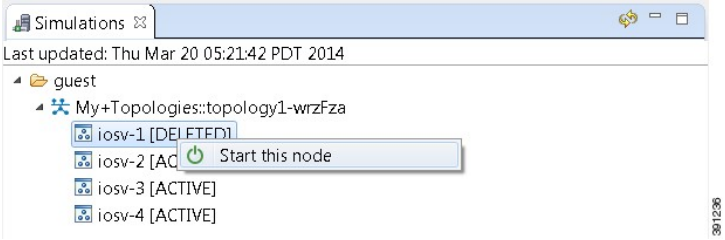
What to Do Next

- (Optional) Start a node.
- (Optional) Stop the simulation.

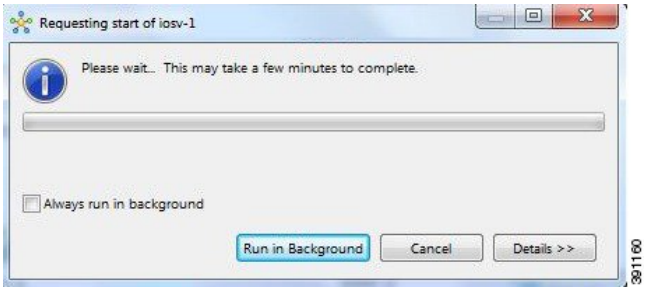
Start a Single Node

Step 1 Right-click a stopped node.
When a node is stopped, its status changes to [DELETED].

Step 2 Click **Start this node**.



The **Requesting start** dialog box appears.



Step 3 Choose one or more of the following actions:

a) Check the **Always run in the background** check box. All future node start requests, stop requests, and simulation launch requests run in the background and do not display dialog boxes.
Note To control the background setting, choose **File** > **Preferences** > **General**.

b) Click **Run in background**.
The dialog box closes while the node simulation stops.

c) Click **Cancel** to return to the **Simulations** view.

d) Take no action and the node simulation restarts momentarily.

Tip When you click **Run in background**, the status bar displays a progress icon. Click the icon to display a compact view of the progress. If an error is encountered, the icon shows a red **X**. Click the error icon to display the error dialog box.

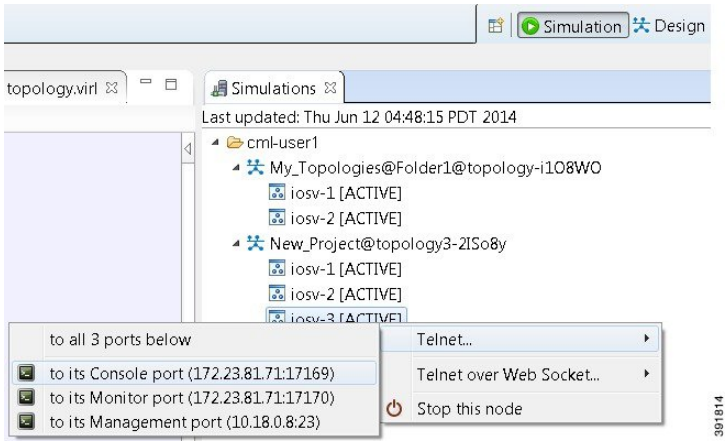
What to Do Next

- (Optional) Stop the simulation.

Modify a Node Configuration in the Simulation

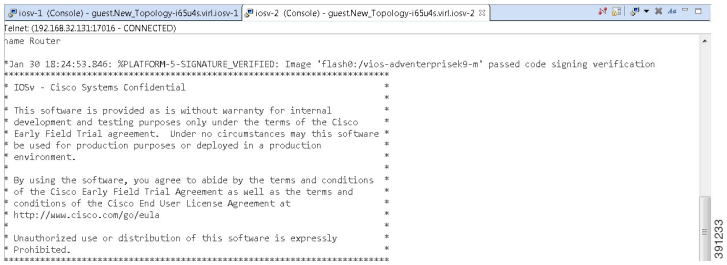
Step 1 Connect to a console on a router node. Right-click the node in the **Simulations** view and choose **Telnet > to its Console port**.

Figure 71: Connecting to a Node Console



A new **Terminal** view opens.

Figure 72: Terminal View



Step 2 If no banner or router prompt is visible, press **Enter**.

You are now working with the operating system running on the node, for example, Cisco IOS software.

Step 3 Use the operating system commands to view or modify the node configuration.

Note Changes you make to the configuration do not appear in the canvas of the **Topology Editor**.

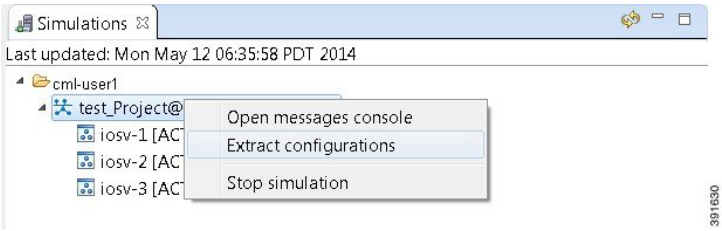
Extract and Save Modified Configurations

Before You Begin

- You have modified a configuration within one or more nodes running within the simulation and want to save the changes.
- Ensure that all routers in the simulation are operational before attempting to extract their configurations.

Step 1 In the **Simulations** view, right-click the topology name, making sure not to click the node name, and select **Extract Configurations**.

Figure 73: Simulations View Context Menu

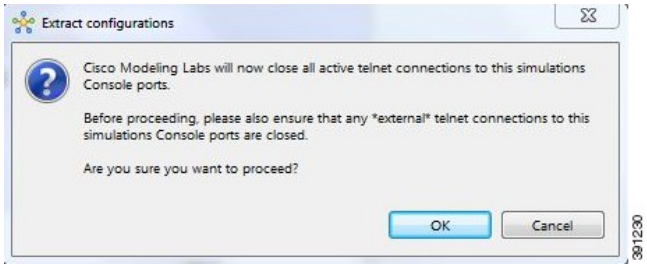


A confirmation dialog box appears.

Step 2 Ensure that all external Telnet connections to the simulation are closed before proceeding.

Note You must close all external Telnet connections to the simulation before you can proceed.

Figure 74: Extract Configurations Dialog Box

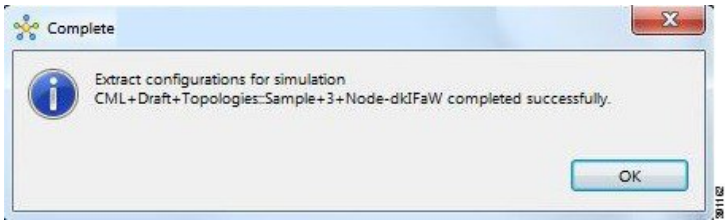


Step 3 Click **OK**.

A dialog box appears.

Step 4 Click **Run in Backgroud**.
When the extraction is complete, a message is displayed.

Figure 75: Extraction Complete Message



Note The configuration information is extracted and saved in the *filename.vir*l file that corresponds to the topology. For example, the file New_Topology that is running as a New_Topology-dkIFaW simulation has its configuration extracted to the file **New_Topology.virl**.

Step 5 Click **OK**.

What to Do Next

- (Optional) Stop the simulation.
- (Optional) Stop a single node in the simulation.

CHAPTER 7

External Connectivity in Cisco Modeling Labs

External Connectivity to a Node

Depending on how the Cisco Modeling Labs environment has been set up by the system administrator, you may have several ways to externally connect to the nodes in a running simulation.

1. Bypass the Cisco Modeling Labs client and connect directly to nodes via Out of Band Management IP access using FLAT

   If enabled by the system administrator, when designing your virtual network, you can specify that you want all nodes to be configured on a reserved management network. All management interfaces are connected to a shared management network segment known as FLAT. This set up will allow you to bypass the Cisco Modeling Labs client and connect directly via Telnet to the nodes.

2. Connecting to external devices using FLAT Inband access

   When enabled, you can configure your virtual topologies to connect and pass data-plane and control-plane packets to one or more external devices such as routers or traffic generators during a simulation. Since FLAT is a Layer 2 solution, the IP addresses in your topology are reachable externally. The **L2 External FLAT** tool in the Cisco Modeling Labs client GUI is used to enable this option.

Note Your simulation continues to be driven through the Cisco Modeling Labs client GUI by communicating with the Cisco Modeling Labs server at its IP address that is bound to the relevant management port.

3. Connecting to external devices using SNAT Inband access

   As an alternative to Inband access using FLAT, you can set up the Static NAT (SNAT) approach. SNAT is a Layer 3 solution that leverages the use of an internal SNAT router to hide the IP addresses in your topology. This router, internal to the Cisco Modeling Labs server, translates IP addresses inbound and

outbound which means that the addressing schemes used on the virtual topology are not propagated outside the virtual network.

When configured, an internal and an external address are assigned to the SNAT-assigned interface on the nodes. For example, configuring 10.11.12.1 as the internal address, and mapping it to 172.16.2.51 externally. Traffic sent to 172.16.2.51 will be translated to the correct internal address and presented to the appropriate node.

The **L3 External SNAT** tool in the Cisco Modeling Labs client GUI is used to enable this option.

Note Your simulation continues to be driven through the Cisco Modeling Labs client GUI by communicating with the Cisco Modeling Labs server at its IP address that is bound to the relevant management port.

Set Up a FLAT Network for Out of Band (OOB) Management Access

By using an out-of-band (OOB) network connection to the Cisco Modeling Labs simulation, you can connect an external application to the Management port of any router node in the simulated network. All management ports will have an IP address on the OOB network, which is a separate network from the other interface IP addresses.

Before You Begin

- Ensure that you have installed a third-party application for Telnet connections, such as PuTTY.
- Ensure that FLAT connectivity has been enabled by the system administrator.

Step 1 Log in to the Cisco Modeling Labs client.

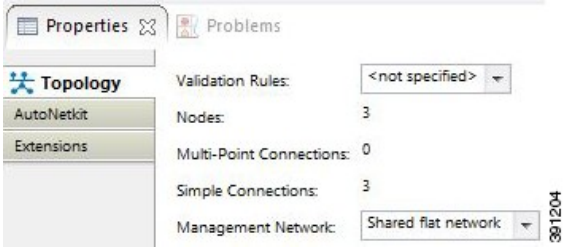
Step 2 Verify you have connectivity to the Cisco Modeling Labs server.

Step 3 Open the **Design** perspective, if it is not already open.

Step 4 Open an existing topology or create your network topology. See the section "Create a Topology" for information on how to do this.

Step 5 Configure the OOB management network.

a) Click on the **Topology Editor** canvas.
b) Choose **Properties > Topology**.
c) Locate **Management Network** and select **Shared flat network** from the drop-down list.



The **Shared flat network** option is what enables the OOB connectivity.

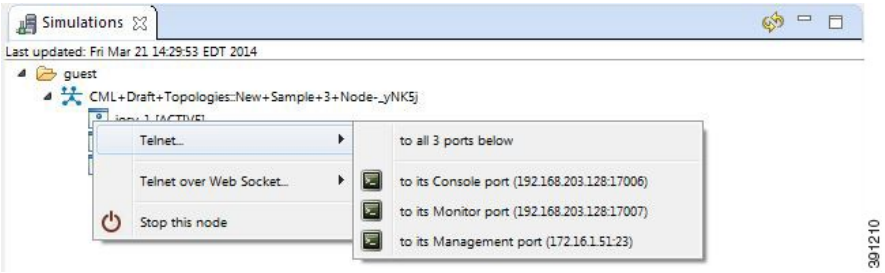
Step 6 From the toolbar, click the **Update Router Configurations** tool.

Note For the purpose of this task, you may retain the default settings.

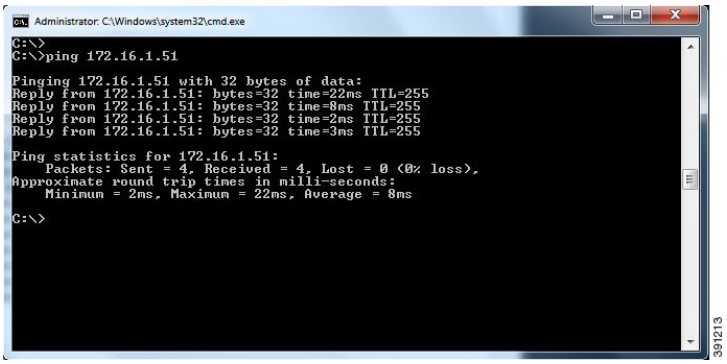
Step 7 After the build phase is completed, initiate the simulation by clicking the **Launch Simulation** tool in the toolbar, and then change to the **Simulation** perspective.

Step 8 (Optional) Enter the **ping** command on the PC to confirm that the management port on a running node can be reached.

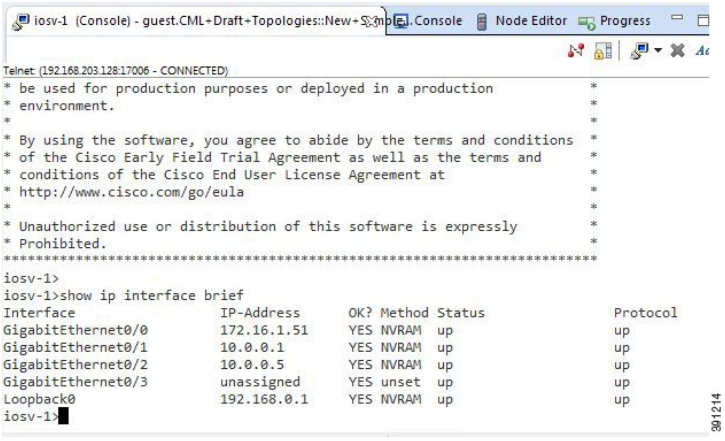
a) In the Cisco Modeling Labs client simulation, select a router node and display the IP address for its management port. In this example, the IP address for the node management port is 172.16.1.51.



b) In the PC command window, enter the **ping** command to target the node management port identified in Step 8a.



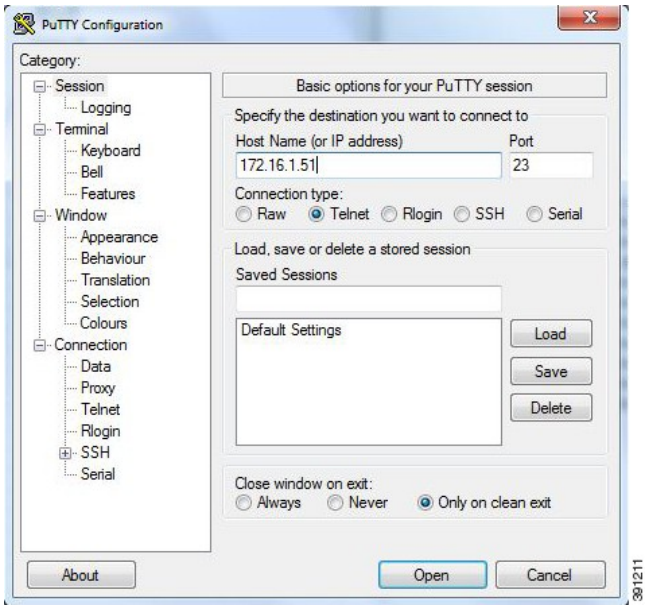
Step 9 (Optional) In the Cisco Modeling Labs client simulation, connect to a console port on a node and confirm the management network IP address exists. Enter the **show ip interface brief** command to display the IP addresses. For this example, use the IP address 172.16.1.51.



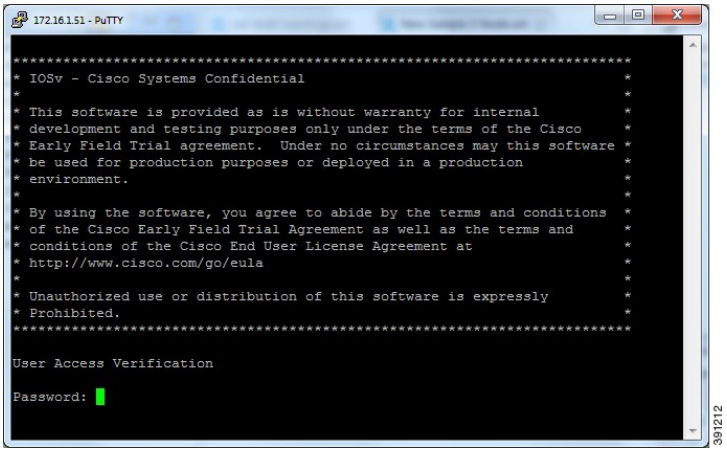
Step 10 Launch and configure a terminal emulator for a Telnet connection to the node management port. In this example, the PuTTY application is used.

a) Launch the PuTTY application.

b) In the **PuTTY Configuration window**, enter the IP address for a Telnet connection to the management port. In this example, the IP address for the node management port is 172.16.1.51.



c) When successful, a Telnet window opens and the router console information is displayed.



Set Up a FLAT Network for Inband Access

The following procedure describes how to connect your virtual network topology to physical devices that are external to the Cisco Modeling Labs server environment. In this procedure, you will be configuring a FLAT network. A FLAT network uses Layer 2 connectivity in which the IP address information about your virtual network can be viewed externally.

Before You Begin

- Ensure that the system administrator has configured the Cisco Modeling Labs server to allow FLAT connections.
- Ensure that you obtain the IP address of the default gateway from the system administrator.
- Ensure that you have the IP address of the external device to which the nodes will connect.

Note We recommend that you first draw your intended design, and then label the devices with the appropriate IP addresses.

Step 1 Log in to the Cisco Modeling Labs client.

Step 2 Verify you have connectivity to the Cisco Modeling Labs server.

Step 3 Open the **Design** perspective, if it is not already open.

Step 4 Open an existing topology or create your network topology. See the section "Create a Topology" for information on how to do this.

Step 5 Click on the canvas to open the **Topology** tab in the **Properties** view and ensure that **Management Network** displays **<not specified>**.

Step 6 From the **Palette** view, under **General**, click the **L2 External FLAT** tool, and then click the canvas to add one FLAT cloud network icon to each corresponding node icon on the canvas.

Step 7 From the **Palette** view, under **Tools**, click the **Connect** tool and connect the **L2 External FLAT** to the desired nodes.

Note L2 External FLAT connections can only be assigned to one interface on one node.

Step 8 From the toolbar, click the **Update Router Configurations** tool.

Note For the purpose of this task, you may retain the default settings.

The system automatically assigns an IP address to the node interface that is connected to the FLAT cloud during the **Simulation** phase, not the **Build** phase.

Step 9 After the build phase is completed, start the simulation by clicking the **Launch Simulation** tool in the toolbar, and then change to the **Simulation** perspective.

Step 10 Log in to the node that is connected to the FLAT network when its state has changed to ACTIVE.

Step 11 View the IP address assigned to the FLAT network and determine if it is on the same subnet as the gateway IP address provided by your system administrator. The L2 external address for each node can be viewed from the **Simulations** view or from each individual node.

- If yes, continue to Step 12.
- If no, advise your system administrator and request the correct gateway IP address.

Step 12 If the external device is on a different subnet, define a default gateway or a broadcast domain that points to the IP address of the gateway that the system administrator supplies.

Prior to completing this step, if you do a **show route**, you will see that no default gateway is defined in the node. This is the default set up. This step allows you to inform the node what path to take in order to reach the external environment via the gateway. An alternative to defining a gateway or static route, with IOSv devices, is to enable DHCP on the interface used for external connectivity.

Note Not all virtual images support DHCP. Check the supported features for the virtual images to determine which ones do support DHCP.

Step 13 Test the connection by pinging from the node to the external (physical) device. If required, turn on **debug ip icmp** or use the **tracert** command to see the progress through the network.

Note You must repeat steps 10-13 for each node that is connected to the FLAT network.

If the ping does not work, confirm with your system administrator that the Cisco Modeling Labs server is configured to support FLAT connectivity and that the gateway IP address you have been provided is correct.

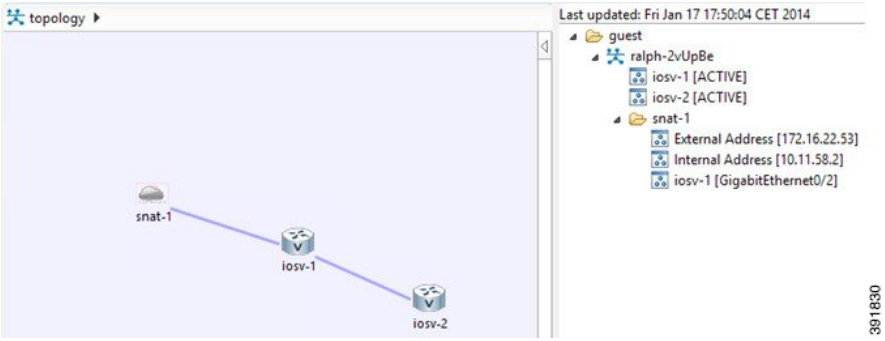
If these items are correct, ping to each of the key devices in the path to determine where the failure occurs and notify your system administrator of the failure source if it is outside your Cisco Modeling Labs environment.

Set Up a SNAT Network for Inband Access

SNAT functionality also allows connected nodes to connect to external devices. Using SNAT, the connectivity occurs at Layer 3 and allows you to hide the simulated virtual network behind a Static NAT address block. The Cisco Modeling Labs server will do a static IP address translation (hence SNAT), translating the private address range allocated to the SNAT 'inside' network into the IP network configured for the SNAT 'outside' network. By default, Cisco Modeling Labs uses an internal gateway function that acts as a SNAT router.

In the example used here, the iosv-1 GE 0/2 interface has two IP addresses. The addresses on the 172.16.22.0 subnet are reachable from the external devices and will be translated to the internal addresses which are on the 10.11.58.0 network.

Figure 76: SNAT Functionality



Using SNAT, the iosv-1 host will have an internal IP address from the 10.11.58.0 infrastructure network on its interface facing the SNAT cloud (10.11.58.2). This will be mapped by the SNAT router for that segment into 172.16.22.53. Assuming that the SNAT interface is connected and there is another host on that segment, such as a gateway, you can now connect to the iosv-1 router via the 172.16.22.53 address.

Note No default gateway is created on the IOSv router. A static route must be added to the iosv-1 router configuration similar to the following example:
ip route 0.0.0.0 0.0.0.0 10.11.58.1

where 0.0.0.0 0.0.0.0 refers to all destination IP addresses not found in the routing table and 10.11.58.1 refers to the SNAT router.

Before You Begin

- Ensure that the system administrator has configured the Cisco Modeling Labs server to allow SNAT connections.
- Ensure that the system administrator provides you with the internal and external IP addresses of the SNAT router.
- Ensure that you have the IP address to the default gateway.
- Ensure that you have the IP address of the external device to which the nodes will connect.

Step 1 Log in to the Cisco Modeling Labs client.

Step 2 Verify you have connectivity to the Cisco Modeling Labs server.

Step 3 Open the **Design** perspective, if it is not already open.

Step 4 Open an existing topology or create your network topology. See the section "Create a Topology" for information on how to do this.

Step 5 Click on the canvas to open the **Topology** tab in the **Properties** view and ensure that **Management Network** displays **<not specified>**.

Step 6 From the **Palette** view, under **General**, click the **L3 External SNAT** tool, and then click the canvas to add one SNAT cloud network icon to each corresponding node icon on the canvas.

Step 7 From the **Palette** view, under **Tools**, click the **Connect** tool and connect the **L3 External SNAT** to the desired nodes.

Note L3 External SNAT connections can only be assigned to one interface on one node.

Step 8 From the toolbar, click the **Update Router Configurations** tool.

Note For the purpose of this task, you may retain the default settings.

The system automatically assigns the internal and external IP addresses to the node interface that is connected to the SNAT cloud during the **Simulation** phase, not the **Build** phase. Therefore, no assigned addresses are visible at this point.

Step 9 After the build phase is completed, start the simulation by clicking the **Launch Simulation** tool in the toolbar, and then change to the **Simulation** perspective.

Step 10 Log in to the node that is connected to the SNAT network when the status of the device has changed to ACTIVE.

Step 11 Create a static route or define a default gateway for the external connection IP addresses that point to the SNAT router's internal IP address. For example, if the external connections are part of the subnet 172.16.2.0/24, and the SNAT router's internal IP address is 10.11.11.1, then the route statement would be:
ip route 172.16.2.0 255.255.255.0 10.11.11.1

An alternative to defining a gateway or static route, with IOSv devices, is to enable DHCP on the interface used for external connectivity.

Note Not all virtual images support DHCP. Check the supported features for the virtual images to determine which ones do support DHCP.

Step 12 Test the connection by pinging from the node to the external (physical) device. If required, turn on **debug ip icmp** or use the **tracert** command to see the connectivity between the end points.

If the ping does not work, confirm with your system administrator that the Cisco Modeling Labs server is configured to support SNAT connectivity and that the gateway IP address you have been provided is correct.

If these items are correct, ping to each of the key devices in the path to determine where the failure occurs and notify your system administrator of the failure source if it is outside your Cisco Modeling Labs environment.