



Cisco Modeling Labs Corporate Edition User Guide, Release 1.1

Last Modified: September 28, 2015

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

THE SPECIFICATIONS AND INFORMATION REGARDING THE PRODUCTS IN THIS MANUAL ARE SUBJECT TO CHANGE WITHOUT NOTICE. ALL STATEMENTS, INFORMATION, AND RECOMMENDATIONS IN THIS MANUAL ARE BELIEVED TO BE ACCURATE BUT ARE PRESENTED WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED. USERS MUST TAKE FULL RESPONSIBILITY FOR THEIR APPLICATION OF ANY PRODUCTS.

THE SOFTWARE LICENSE AND LIMITED WARRANTY FOR THE ACCOMPANYING PRODUCT ARE SET FORTH IN THE INFORMATION PACKET THAT SHIPPED WITH THE PRODUCT AND ARE INCORPORATED HEREIN BY THIS REFERENCE. IF YOU ARE UNABLE TO LOCATE THE SOFTWARE LICENSE OR LIMITED WARRANTY, CONTACT YOUR CISCO REPRESENTATIVE FOR A COPY.

The Cisco implementation of TCP header compression is an adaptation of a program developed by the University of California, Berkeley (UCB) as part of UCB's public domain version of the UNIX operating system. All rights reserved. Copyright © 1981, Regents of the University of California.

NOTWITHSTANDING ANY OTHER WARRANTY HEREIN, ALL DOCUMENT FILES AND SOFTWARE OF THESE SUPPLIERS ARE PROVIDED "AS IS" WITH ALL FAULTS. CISCO AND THE ABOVE-NAMED SUPPLIERS DISCLAIM ALL WARRANTIES, EXPRESSED OR IMPLIED, INCLUDING, WITHOUT LIMITATION, THOSE OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT OR ARISING FROM A COURSE OF DEALING, USAGE, OR TRADE PRACTICE.

IN NO EVENT SHALL CISCO OR ITS SUPPLIERS BE LIABLE FOR ANY INDIRECT, SPECIAL, CONSEQUENTIAL, OR INCIDENTAL DAMAGES, INCLUDING, WITHOUT LIMITATION, LOST PROFITS OR LOSS OR DAMAGE TO DATA ARISING OUT OF THE USE OR INABILITY TO USE THIS MANUAL, EVEN IF CISCO OR ITS SUPPLIERS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

Any Internet Protocol (IP) addresses and phone numbers used in this document are not intended to be actual addresses and phone numbers. Any examples, command display output, network topology diagrams, and other figures included in the document are shown for illustrative purposes only. Any use of actual IP addresses or phone numbers in illustrative content is unintentional and coincidental.

Cisco and the Cisco logo are trademarks or registered trademarks of Cisco and/or its affiliates in the U.S. and other countries. To view a list of Cisco trademarks, go to this URL: <http://www.cisco.com/go/trademarks>. Third-party trademarks mentioned are the property of their respective owners. The use of the word partner does not imply a partnership relationship between Cisco and any other company. (1110R)

© 2015 Cisco Systems, Inc. All rights reserved.



CONTENTS

Preface

Preface ix

Document Conventions ix

Related Documentation xi

Obtaining Documentation and Submitting a Service Request xi

CHAPTER 1

Overview of Cisco Modeling Labs 1

Cisco Modeling Labs 1

Cisco Modeling Labs Server Components 1

Cisco Modeling Labs Client 2

Virtual Images 2

Cisco Modeling Labs Component Requirements 3

Cisco Modeling Labs Framework 5

CHAPTER 2

Using the Cisco Modeling Labs Client 7

Using the Cisco Modeling Labs Client Overview 7

Navigating Within the Cisco Modeling Labs Client 7

Menu Bar 12

Importing a Topology File 16

Exporting a Topology File 16

Generate Problem Reports 17

Toolbar 17

Cisco Modeling Labs Client Components 18

Cisco Modeling Labs Client Editors 19

Topology Editor 19

Node Editor 19

Cisco Modeling Labs Client Perspectives 20

Working with Perspectives 21

Customize Perspectives	22
Design Perspective	23
Simulation Perspective	24
Cisco Modeling Labs Client Views	24
Console View	26
Graph Overview	27
History View	28
Outline View	30
Palette View	31
Problems View	33
The Quick Fix Option	36
Projects View	38
Properties View	39
Node Properties	41
Topology Properties	46
Interface Properties	51
Connection Properties	52
Search View	52
Simulations View	55
Topology Options	58
Node Options	60
Terminal View	61
Setting Preferences for the Cisco Modeling Labs Client	62
Node Subtypes Setting	62
Fetch Node Subtypes from the Cisco Modeling Labs Server	64
Terminal Setting	64
Topology Editor Setting	69
Web Services Setting	70
AutoNetkit Visualization Setting	72
Web Browser Setting	75
Secure Storage Setting	76
Resetting the Secure Storage Password	77

CHAPTER 3**Design a Topology 79**

Design a Topology Overview	79
----------------------------	----

Topology Nodes and Connections	79
Create a Topology	82
Method 1: Create a Topology from the Menu Bar	82
Method 2: Create a Topology from the Projects View	83
Method 3: Create a Topology from the Toolbar	83
Place the Nodes on the Canvas	83
Create Connections and Interfaces	84
Use Unmanaged Switches	85
The Cisco IOSvL2 Switch Image	86
Use the Cisco IOSvL2 Switch Image	87
Create a Site	90

CHAPTER 4**Build a Configuration 93**

Build a Configuration Overview	93
Create and Modify a Node Configuration	93
Create a Node Configuration Manually	94
Use an Existing Node Configuration	95
Import the Configuration from a Cariden MATE File	95
Export the Configuration to Cariden MATE File	98
Import the Configuration from a Visio vsdx File	99
Export the Configuration to SVG Files	102
Create Node and Interface Configurations Using AutoNetkit	102
Assign VLANs	104
Use a Managed Switch	105
Use Multiple Managed Switches	107
Use Multiple Unconnected Managed Switches	109
Set Firewall Capabilities	110
Set Security Levels	112
Configure GRE Tunnels	114

CHAPTER 5**Visualizing the Topology 121**

AutoNetkit Visualization	121
Access AutoNetkit Visualization	123
AutoNetkit View Options	125

CHAPTER 6**Simulate the Topology 131**

- Simulate the Topology Overview 131
- Launch a Simulation 132
 - Jumphost Virtual Machine (VM) 136
 - Linux Container (LXC) 137
 - Static Port Assignment to the LXC 138
 - Launch a Phased Simulation 139
 - Launch a Time-Limited Simulation 143
 - Reset the Time Limit on a Running Simulation 144
 - Control Interface States 145
- Connect to a Simulation Node Console 148
 - Connect to a Simulation Node Console via SSH 148
 - Connect to Multiple Simulation Node Consoles 151
 - Terminal Multiplexer Functionality 152
- Start a Single Node 154
 - Start a Node in a Running Simulation 156
 - Start Multiple Nodes in a Running Simulation 158
- Stop a Simulation 160
 - Stop a Simulation from the Toolbar 160
 - Stop a Simulation from the Simulations View 161
 - Stop Multiple Simulations from the Simulations View 163
- Stop a Single Node 164
 - Stop Multiple Nodes 165
- Modify a Node Configuration in the Simulation 167
 - Modify a Node Configuration in the Simulation via SSH 168
 - Modify a Node Configuration in the Simulation via Telnet 170
 - Modify Multiple Node Configurations in the Simulation 171
- Extract and Save Modified Configurations 172
 - Partial Configuration Extraction 174
- Linux Server Snapshot Support 175
 - Reuse the Image Snapshot 177

CHAPTER 7**External Connectivity in Cisco Modeling Labs 179**

- External Connectivity to a Node 179

Set Up a FLAT Network for Out of Band (OOB) Management Access **180**
 Add an Additional FLAT Network to a Simulation **183**
Set Up a FLAT Network for Inband Access **186**
Set Up a SNAT Network for Inband Access **188**



Preface

- [Document Conventions](#), page ix
- [Related Documentation](#), page xi
- [Obtaining Documentation and Submitting a Service Request](#), page xi

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 .
<i>Italic font</i>	Document titles, new or emphasized terms, and arguments for which you supply values are in <i>italic font</i> .
Courier font	Terminal sessions and information the system displays appear in <code>courier font</code> .
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, refer to the Cisco Modeling Labs release notes.

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, page 1](#)
- [Cisco Modeling Labs Server Components, page 1](#)
- [Cisco Modeling Labs Client, page 2](#)
- [Virtual Images, page 2](#)
- [Cisco Modeling Labs Component Requirements, page 3](#)
- [Cisco Modeling Labs Framework, page 5](#)

Cisco Modeling Labs

Cisco Modeling Labs 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. It also permits access to the Cisco Modeling Labs server functionality.

Cisco Modeling Labs Server Components

The Cisco Modeling Labs server is available as:

- A Linux distribution that is bundled within the VMware Open Virtual Appliance (OVA) file for VMware ESXi. The bundle includes all the supporting files.
- A Linux distribution that is bundled with an ISO image file for bare-metal environments. The bundle includes all the supporting files.

The Cisco Modeling Labs 1.1 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 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 (VM).
- **Services Topology Director:** Generates OpenStack calls for the creation of VMs and links based on the XML topology definition created by the Cisco Modeling Labs client. Additionally, it provides the bootstrap configuration, which can be autogenerated, 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 use a graphical point-and-click editor to quickly create and edit complex network topologies in a sandbox.
- 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 1.1 includes the following images built into the Cisco Modeling Labs client:

- Cisco Virtual IOS (IOSv) Software Release 15.5(3)M
- Cisco IOSv Layer 2 Switch Software Release 15.2 (4.0.63)E
- Linux server (Ubuntu 14.04.2 LTS Cloud-init)

Additionally, the following demonstration images are included in the build:

- Cisco IOS XRv Software Release 5.3.1 CCO
- Cisco CSR1000v Software Release 3.1.6 XE-based

The following production images are available to purchase separately:

- Cisco IOS XRv Software Release 5.3.1
- Cisco CSR1000v Software Release 3.1.6

- Cisco ASAv Software Release 9.4.1

See [Release Notes for Cisco Modeling Labs 1.1](#) for more information on Cisco virtual software-supported features.

Cisco Modeling Labs Component Requirements

The following tables identify the minimum requirements for installing and operating the Cisco Modeling Labs Corporate Edition, Release 1.1. For additional information, see [Cisco Modeling Labs Corporate Edition Client Installation Guide, Release 1.1](#) and [Cisco Modeling Labs Corporate Edition System Administrator Installation Guide, Release 1.1](#). These guides provide detailed information on platform requirements for installing and operating Cisco Modeling Labs Corporate Edition, Release 1.1 for system administrators and end users.



Important

The requirements listed in the following table are for topologies containing Cisco IOSv nodes only. You must use the resource calculator for topologies containing nodes other than Cisco IOSv nodes.

This is a basic rule of thumb since there are many variables, such as the type of nodes running (Cisco IOSv versus Cisco IOS XRv), the number of routing protocols, the timers set on each node, the amount of traffic, and so on. Each of these use their share of computational resources that are usually running on dedicated hardware.

Table 1: Server Hardware Requirements

Requirement	Description
Small Installation (Minimum)	Server with capacity to run base package of 15 IOSv nodes
Memory (RAM)	16 GB
Disk Space	250 GB minimum
Processors	4 CPU cores
Small and Medium Installation	Server with a capacity to run up to 50 nodes
Memory (RAM)	128 GB
Disk Space	250 GB minimum
Processors	16 CPU cores
Large Installation	Server with a capacity to run up to 100 nodes
Memory (RAM)	256 GB
Disk Space	250 GB minimum
Processors	40 CPU cores

Table 2: Server Software Requirements

Requirement	Description
VMware vSphere	<p>One of the following:</p> <ul style="list-style-type: none"> • ESXi 5.1 U2 (Build 1483097) • ESXi 5.5 U1 (Build 1623387) • ESXi 6.0 (Build 2494585) <p>Note You must verify that you are using vSphere Client v5.5 Update 2 (Build 1993072) or later before deploying Cisco Modeling Labs. Failure to use the minimum version will result in a failed deployment that returns an error stating that nested virtualization is not supported.</p>
Browser	<p>One of the following:</p> <ul style="list-style-type: none"> • Google Chrome Version 33.0 or later • Internet Explorer 10.0 or later • Mozilla Firefox 28.0 or later • Safari 7.0 or later <p>Note Internet Explorer is not supported for use with the AutoNetkit Visualization functionality and the User Workspace Management interface.</p>

Table 3: Client Hardware Requirements

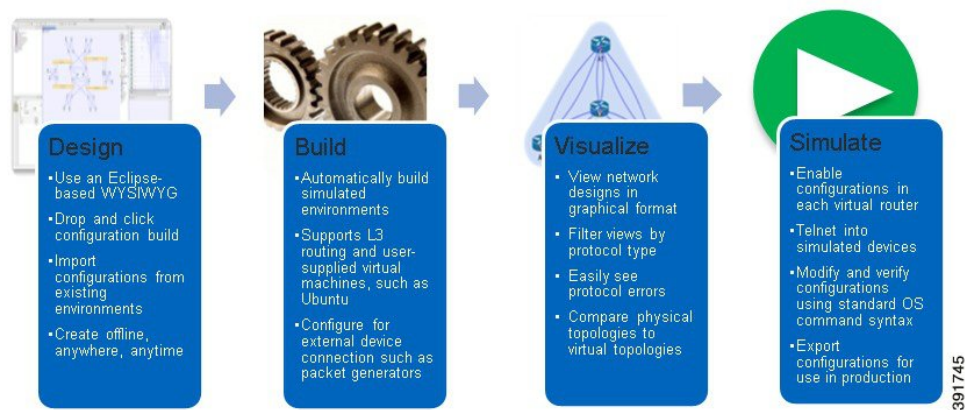
Requirement	Description
Operating System	<p>Either of the following:</p> <ul style="list-style-type: none"> • Microsoft Windows <ul style="list-style-type: none"> ◦ Windows 7 or ◦ Windows 8 • OS X <ul style="list-style-type: none"> ◦ OS X 10.8 or later
Memory (RAM)	500 MB
Disk Space	200 MB

Table 4: Client Software Requirements

Requirement	Description
Java Runtime Environment (JRE)	For Windows and OS X, either of the following: <ul style="list-style-type: none"> • Version 6 • Version 7 • Version 8
Browser	One of the following: <ul style="list-style-type: none"> • 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 <p>Note Internet Explorer is not supported for use with the AutoNetkit Visualization functionality and the User Workspace Management interface.</p>

Cisco Modeling Labs Framework

Figure 1: Cisco Modeling Labs Framework



Cisco Modeling Labs 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 that are 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 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 IS-IS 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 use Telnet or SSH to connect to the consoles 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: 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 save them and use them as reference when configuring the production network.



Using the Cisco Modeling Labs Client

- [Using the Cisco Modeling Labs Client Overview, page 7](#)
- [Navigating Within the Cisco Modeling Labs Client, page 7](#)
- [Cisco Modeling Labs Client Components, page 18](#)
- [Setting Preferences for the Cisco Modeling Labs Client, page 62](#)

Using the Cisco Modeling Labs Client Overview

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

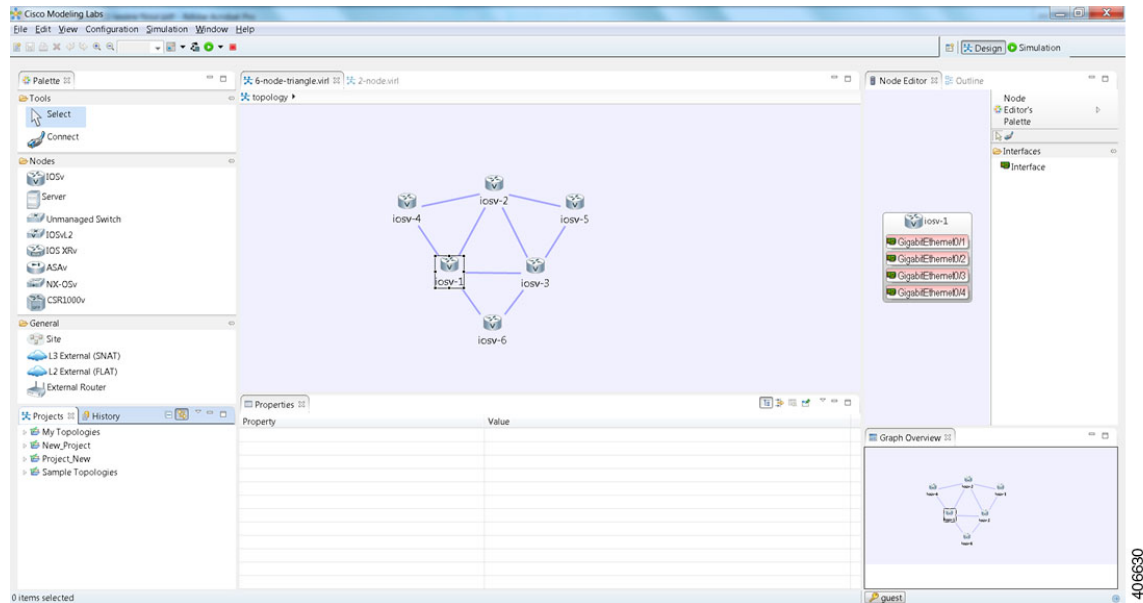
This chapter introduces the main areas and capabilities of the Cisco Modeling Labs client.

Navigating Within the Cisco Modeling Labs Client

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

- **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.

Figure 2: Cisco Modeling Labs Workbench



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

Figure 3: Menu Bar



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

Figure 4: Toolbar



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

Figure 5: Perspectives



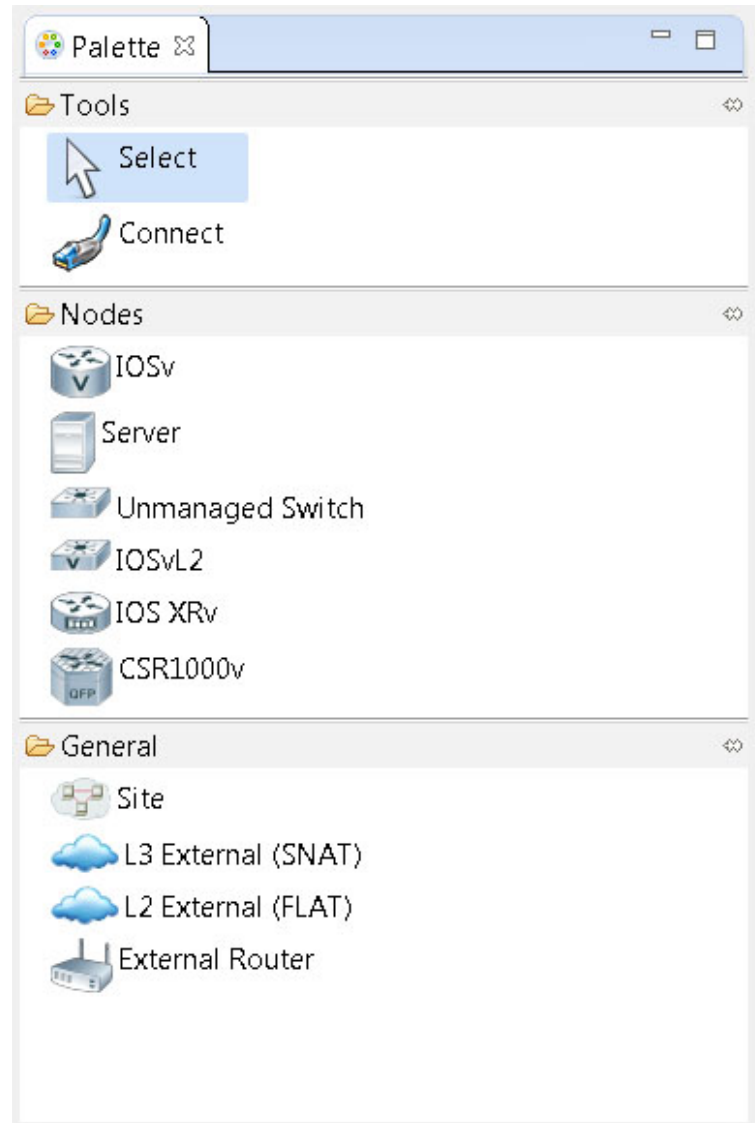
- From the **Design** perspective, you can design your network topology, build the node configurations, and check routing protocols. The **Design** perspective is the default perspective if you are launching the Cisco Modeling Labs client for the first time.
 - From the **Simulation** perspective, you can enable devices and modify configurations to run the simulations. When the nodes in your topology are fully initialized, you can connect to the consoles as you would connect to a router console.
- **Cisco Modeling Labs Client 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**.

Figure 6: Node Editor



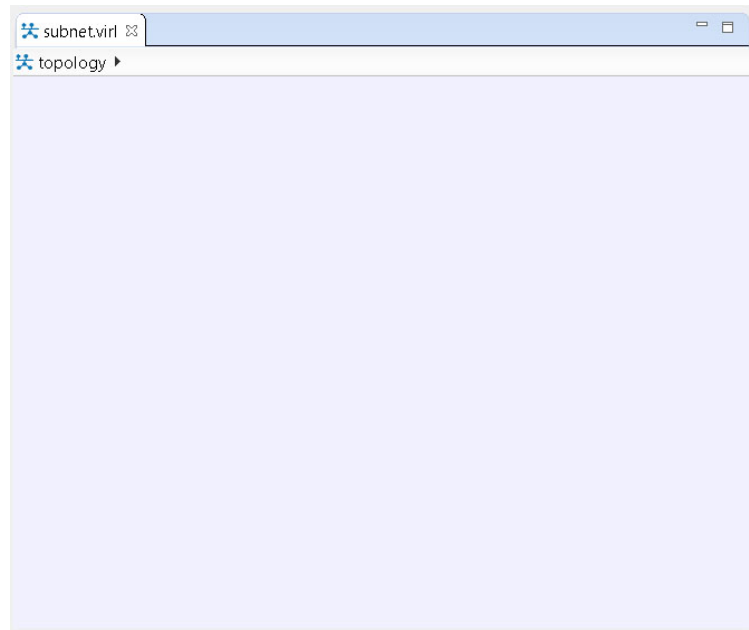
- The **Topology Editor** comprises the **Palette** view

Figure 7: Palette



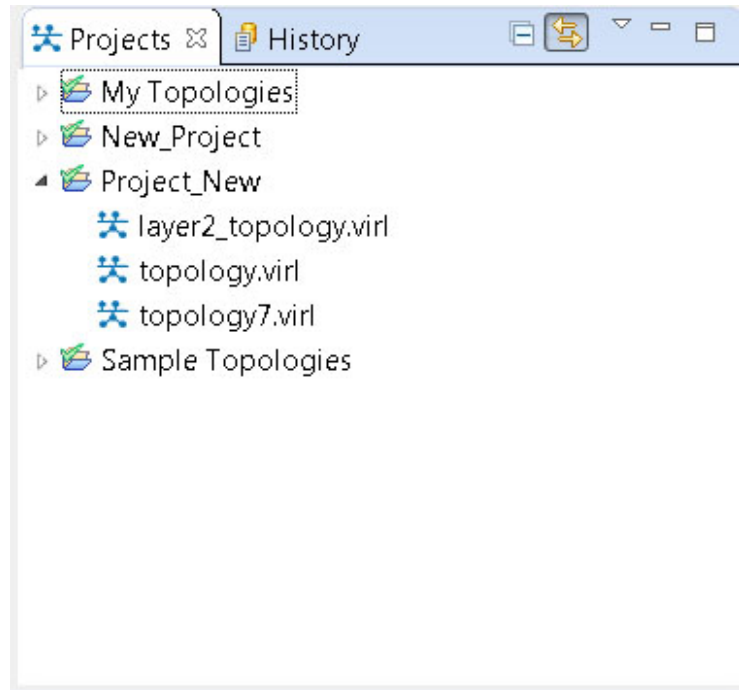
and the canvas.

Figure 8: Canvas



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

Figure 9: Views



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

Menu Bar

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

Table 5: Menu Bar Items

Menu	Action(s)
File	<p>Enables you to perform actions on a topology .virl file, set preferences for resources, and exit the Cisco Modeling Labs client.</p> <ul style="list-style-type: none"> • New: Creates a new topology project folder, topology project, or topology .virl file. See Design a Topology Overview, on page 79 for more information. • Save: Saves the current Topology Editor contents. • Save As: Saves the current Topology Editor contents to a new file. • Save All: Saves all 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 16 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 16 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 62 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	<p>Enables you to manipulate resources on the canvas.</p> <ul style="list-style-type: none"> • 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. • Search: Enables you to perform topology searches and text searches within work spaces, topology projects, files, and working sets. • Select All: Selects all objects in the current topology canvas. • Grid: Provides three options for displaying nodes on the canvas: Display Grid, Snap to Grid, and Distribute Nodes. • Reset Node Subtype: Enables you to redefine a virtual node on the canvas. When Reset All Interface Names is selected, the interface IDs are reset sequentially, starting from the minimum range. • Distribute Nodes: Aligns the nodes on the canvas evenly, either vertically or horizontally. (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. • Ungroup Site: Ungroups nodes from within a site, removing one layer of nesting.

Menu	Action(s)
View	<p>Enables you to manipulate the Cisco Modeling Labs client.</p> <ul style="list-style-type: none"> • Zoom In: Allows you to zoom in on the canvas, increasing the topology view. This is also available on the toolbar. • Zoom Out: Allows you to zoom out on the canvas, reducing the topology view. This is also available on the toolbar. • Highlight Connection: Highlights all connections to and from the selected node. • Show/Hide Map Background: Displays a map background for the topology. Choose Hide to disable the feature. (You must have Internet access to use this feature.) • Topology Labels: Displays the IPv4 and IPv6 loopback addresses on the canvas. Click Show Topology Labels to turn on this functionality. Toggle between Node loopback IPv4 and Node loopback IPv6 to see the applicable addresses on the node(s) on the canvas.
Configuration	<p>Enables you to generate node configurations.</p> <ul style="list-style-type: none"> • Build Initial Configurations: Creates initial node configurations.
Simulation	<p>Enables you to start and stop simulations.</p> <ul style="list-style-type: none"> • Launch Simulation: Starts a simulation. • Launch Simulation (with time limit) : Starts a simulation for a specified time period. • Stop Simulations: Stops a running simulation.
Window	<p>Enables you to manage perspectives in the Cisco Modeling Labs client.</p> <ul style="list-style-type: none"> • Open Perspective: Opens the selected perspective. • Show View : Displays the selected view. • Reset Perspective: Resets the perspective to the initial login perspective. • Save Perspective As: Saves the current perspective to a specified name. • Close Perspective: Closes the open perspective.

Menu	Action(s)
Help	<p>Displays the help topics for using the Cisco Modeling Labs client in a separate browser window.</p> <ul style="list-style-type: none"> • Help Contents: Opens the Cisco Modeling Labs client support documentation. • Cheat Sheets: Helps you in completing 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 Generate Problem Reports, on page 17 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.
-

Generate Problem Reports

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

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



Note

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



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








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

Toolbar

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

Table 6: Toolbar Options

Icon	Function	Description
	New Topology File	Creates a new topology file. A topology file is a .virl file where the network arrangement is designed.
	Save	Saves the current topology.

Icon	Function	Description
	Print	Prints the current topology.
	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.
	Show Topology Labels	Displays either IPv4 or IPv6 loopback addresses configured manually or by AutoNetkit for the topology on the canvas.
	Build Initial Configurations	Generates initial node configurations for the topology.
	Launch Simulation	Launches a simulation indefinitely or for a user-specified time period.
	Stop Simulations	Stops all running simulation(s).

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 19
- [Cisco Modeling Labs Client Perspectives](#), on page 20
- [Cisco Modeling Labs Client Views](#), on page 24

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 **Palette** view and the canvas.
- **Node Editor:** Shows the interface details for the currently selected elements in the **Topology Editor**.

**Note**

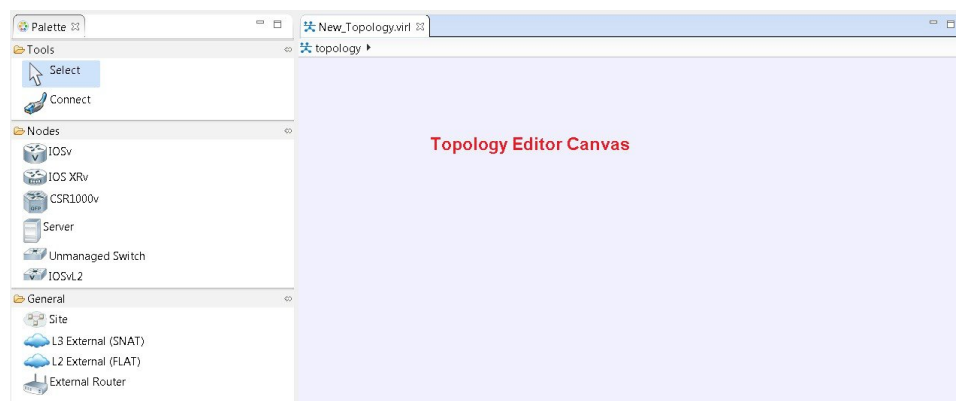
Selecting a connection in the **Topology Editor** shows the details of both the endpoints in the **Node Editor**.

Topology Editor

The **Topology Editor** allows you to:

- Add, move, group, rename, 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 10: Topology Editor



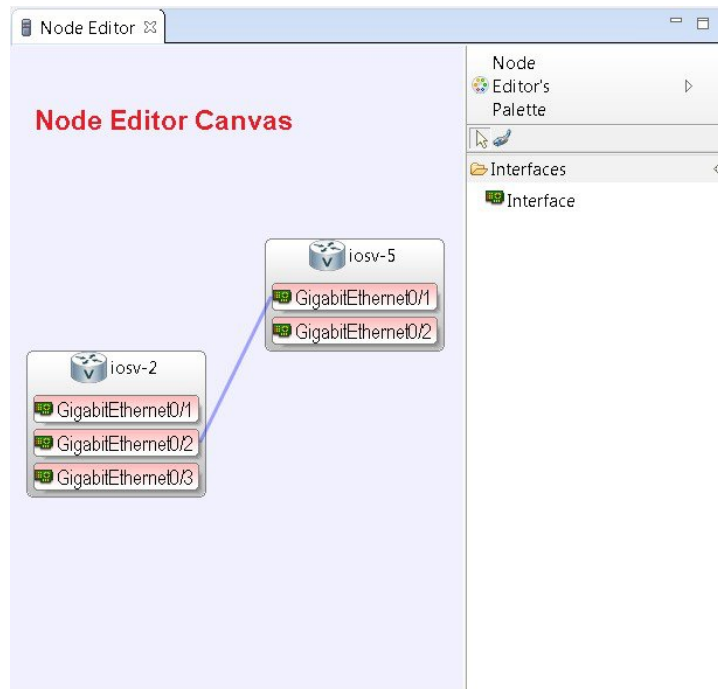
Node Editor

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 11: 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 Cisco IOSv versions.



Note

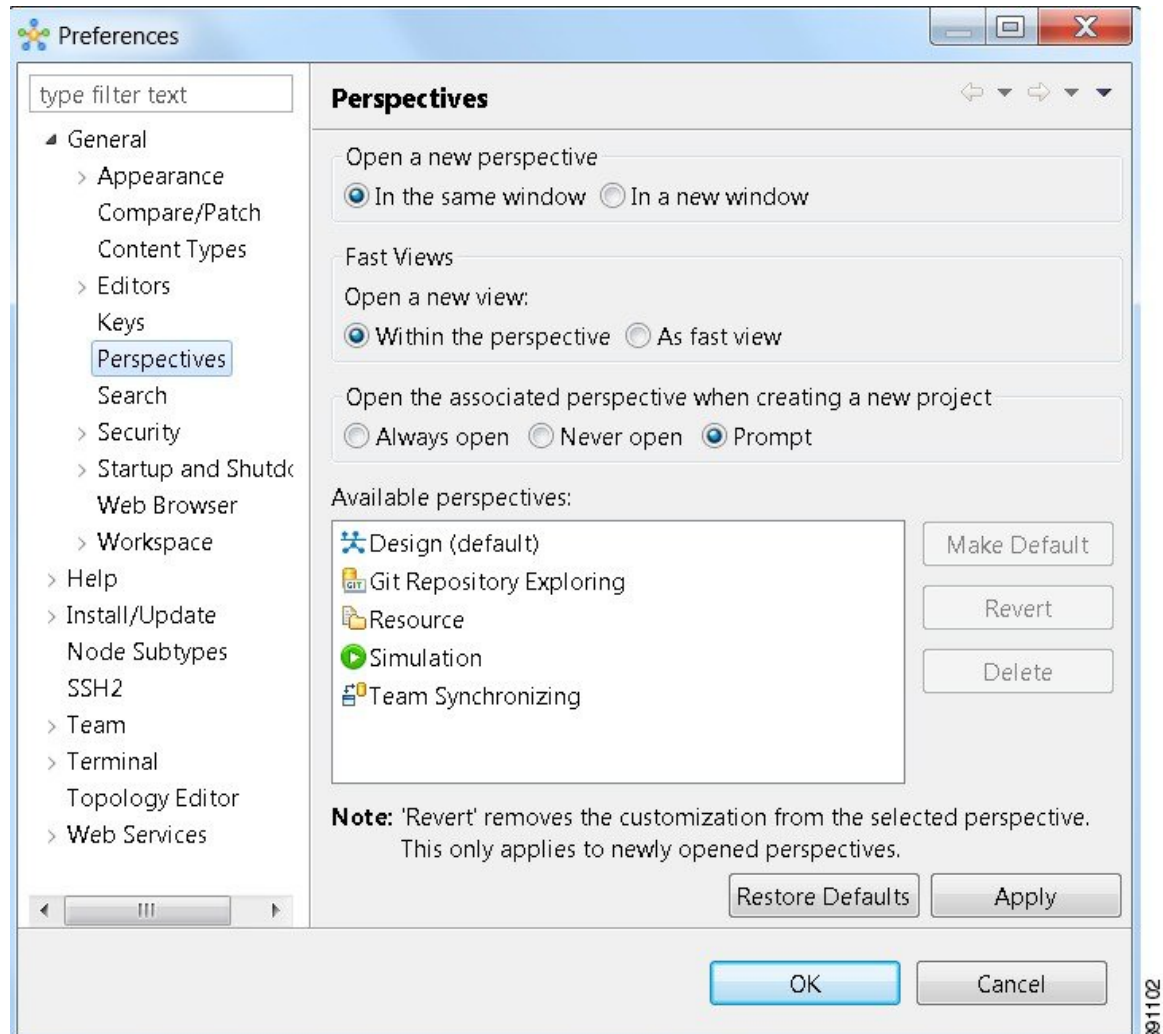
The **Design** and **Simulation** perspectives that are built into the Cisco Modeling Labs client cannot be deleted.

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

Figure 12: Perspectives Preferences



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

Table 7: 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.

Option	Description
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 8: Available Perspectives Options

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

Customize Perspectives

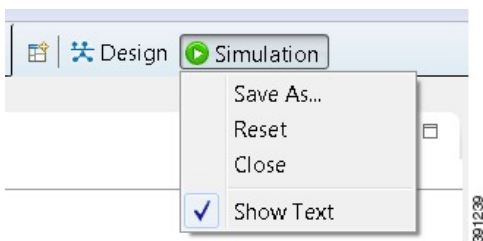
Cisco Modeling Labs provides two perspectives for use in the workspace: the **Design** perspective and the **Simulation** perspective. These perspectives display various views and settings that cannot be changed, nor can they be deleted from the Cisco Modeling Labs client. You can, however, customize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

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



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

Step 1 Open the **Design** perspective or **Simulation** perspective in your workspace.



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

- Step 2** To add new views, choose **Window > Show View > Other** and the **Show View** dialog box is displayed listing all views for use.
Views already in use are shown as dimmed in the list.
- Step 3** 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.
- **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, right-click the **Design** perspective button, select **Save As**, and enter a name for the new **Design** perspective.

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

Table 9: 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.

Operation	Description
Close	Closes the current perspective. You can reopen it using the Open Perspective tool, which is located at the upper right corner of the window.
Show Text	When selected, shows perspective names in the toolbar instead of icons. When deselected, shows perspective icons in the toolbar instead of names.

Simulation Perspective

The **Simulation** perspective opens after you launch a simulation; you are prompted to switch to the **Simulation** perspective. Switching to the **Simulation** perspective means that you can now connect to your running nodes in the **Simulations** view. By default, it incorporates the **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 10: 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	When selected, shows perspective names in the toolbar instead of icons. When deselected, shows perspective icons in the toolbar instead of names.

Cisco Modeling Labs Client Views

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

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

Table 11: Commonly Used Views in the Cisco Modeling Labs Client

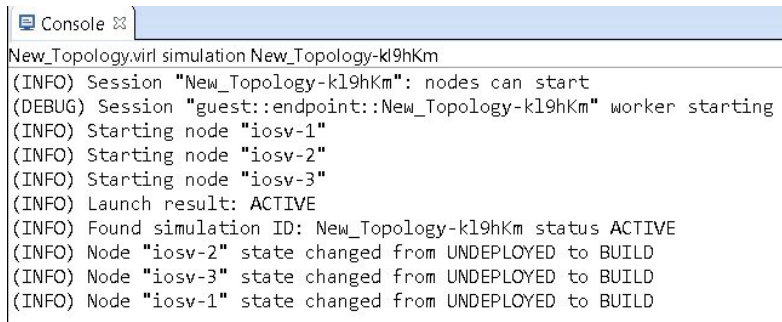
View Name	Perspective Where Used	Description
Console view	Design and Simulation	Design —Displays messages from AutoNetkit when it is used to generate router configurations. Simulation —Displays message streams from the Cisco Modeling Labs server after a simulation is launched. For more information, see Console View , on page 26.
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 27.
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 28.
Outline view	Design	Shows a list of the nodes in the topology and their interface connections. For more information, see Outline View , on page 30.
Palette view	Design	Allows you to add devices and interface connections to your topology. For more information, see Palette View , on page 31.
Problems view	Design	Displays errors, warnings, and other information that was detected within the Topology Editor . For more information, see Problems View , on page 33.
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 38.
Properties view	Design	Displays names and properties of nodes and interfaces. For more information, see Properties View , on page 39.
Search view	Design	Displays the results of a search, which can be based on text strings, regular expressions, patterns, whole words, and case-sensitive characters. For more information, see Search View , on page 52.
Simulations view	Simulation	Displays information on all running simulations. For more information, see Simulations View , on page 55.

View Name	Perspective Where Used	Description
Terminal view	Simulation	Displays console information when you use Telnet or SSH to connect to a node. For more information, see Terminal View , on page 61.

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 13: Console View



```

Console x
New_Topology.virl simulation New_Topology-k19hKm
(INFO) Session "New_Topology-k19hKm": nodes can start
(DEBUG) Session "guest::endpoint::New_Topology-k19hKm" worker starting
(INFO) Starting node "iosv-1"
(INFO) Starting node "iosv-2"
(INFO) Starting node "iosv-3"
(INFO) Launch result: ACTIVE
(INFO) Found simulation ID: New_Topology-k19hKm status ACTIVE
(INFO) Node "iosv-2" state changed from UNDEPLOYED to BUILD
(INFO) Node "iosv-3" state changed from UNDEPLOYED to BUILD
(INFO) Node "iosv-1" state changed from UNDEPLOYED to BUILD

```




391087





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

Figure 14: Console View Toolbar



Table 12: 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.

Icon	Function	Description
	Display Selected Console	Displays the Console view for the selected simulation.
	Open Console	Opens a new Console view.
	Minimize	Reduces the size of the Console view.
	Maximize	Increases the size of the Console view.

**Note**

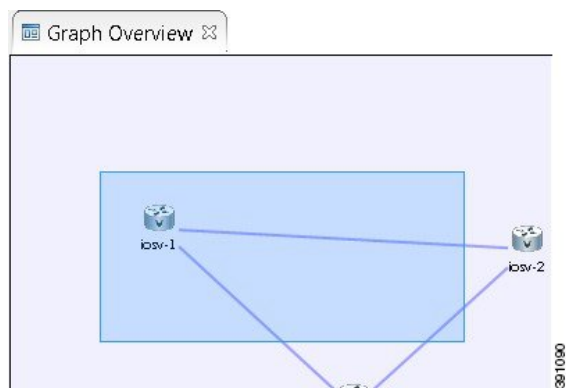
When several simulations are running, use the toolbar button **Display Selected Console** to toggle between the **Console** views for the different simulations.

Graph Overview

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

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

Figure 15: 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**.

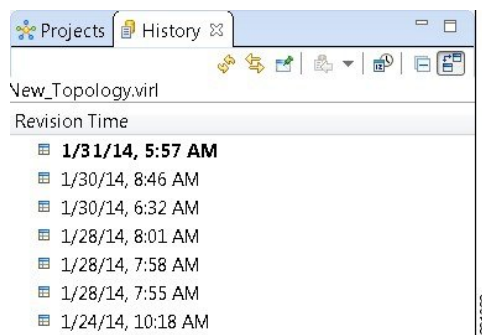
History View

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 16: History View



To view the changed history of a file, in the **Projects** view, right-click the applicable file and choose **Team > Show Local History**.

The **History** view displays a list of revision times; the most recent revision time is highlighted at the top of the list.









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

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

Figure 17: History View Toolbar



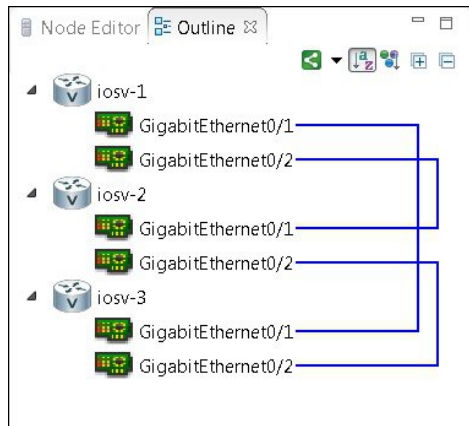
Table 13: Available Tools

Icon	Function	Description
	Refresh	Refreshes the contents of the view, retrieving the latest history information for a file from the system.
	Link with Editor and Selection	Toggles when the History view selection is linked to the active editor. When this option is selected, changing the active editor automatically updates the History view selection to the project, folder, and file being edited.
	Pin this History View	Pins the view to the workbench and captures a snapshot of the file history information. New requests for file history are opened in a new instance of the History view.
	Group Revisions by Date	Sorts all history items by date. Options are: <ul style="list-style-type: none"> • 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 18: Outline View








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

Figure 19: Outline View Toolbar



Table 14: Available Tools

Icon	Function	Description
	Change Display Mode of Connection	Allows you to choose various options when displaying connections: <ul style="list-style-type: none"> • 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.

Icon	Function	Description
	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.

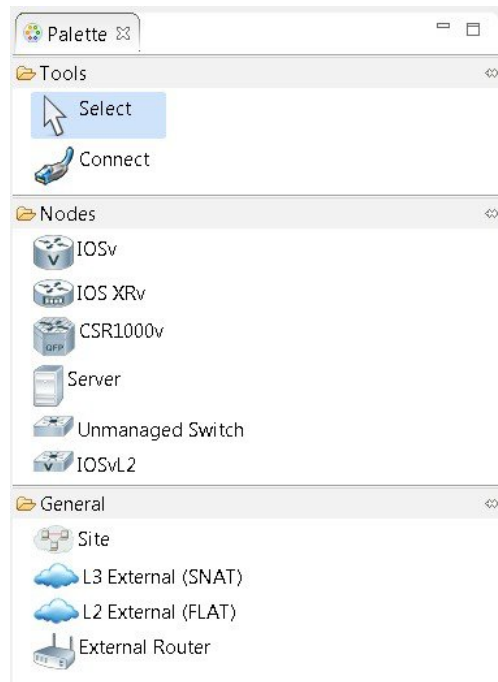
Palette View

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

- Add nodes, Layer 3 external Secure Network Address Translation (SNAT) connections, and Layer 2 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 20: 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, Layer 3 external (SNAT) connections, and Layer 2 external (FLAT) connections on the **Topology Editor** canvas. The **Connect** tool creates connections between node interfaces.



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

- **Nodes:** Contains the node types available for use in topologies. Currently, Cisco Modeling Labs, Release 1.1 includes the following:
 - Cisco IOSv Software Release 15.4(3)M
 - Cisco IOSv Layer 2
 - Ubuntu 14.04 Cloud



Note Additional node subtypes can be installed separately. See [Release Notes for Cisco Modeling Labs Release 1.1](#) for the most up-to-date list of supported virtual images.

- **General:** Contains the different types of connection functions that are supported for nodes. Options are:
 - Layer 3 external connections
 - Layer 2 external connections



You can also create sites.

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

Figure 21: Palette View Toolbar



Table 15: 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, and other information 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.

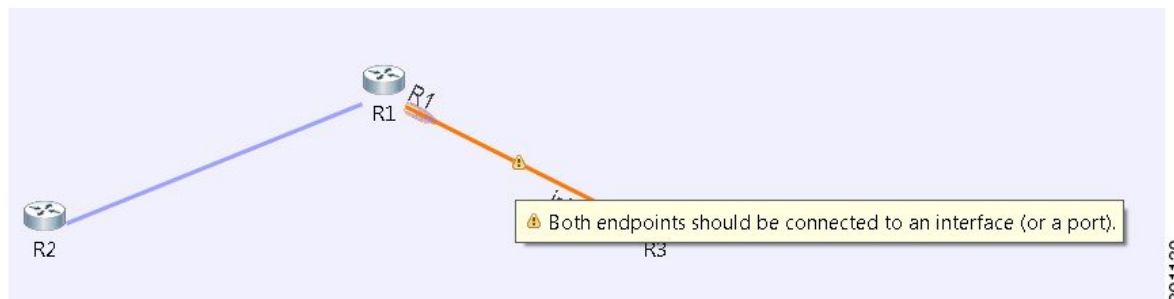
Figure 22: Problems View

Description	Resource	Path	Location	Type
Errors (1 item)				
Warnings (3 items)				
Infos (11 items)				

391104

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 23: Problem Example

391130

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

Figure 24: Problems View Toolbar

Table 16: Available Tools

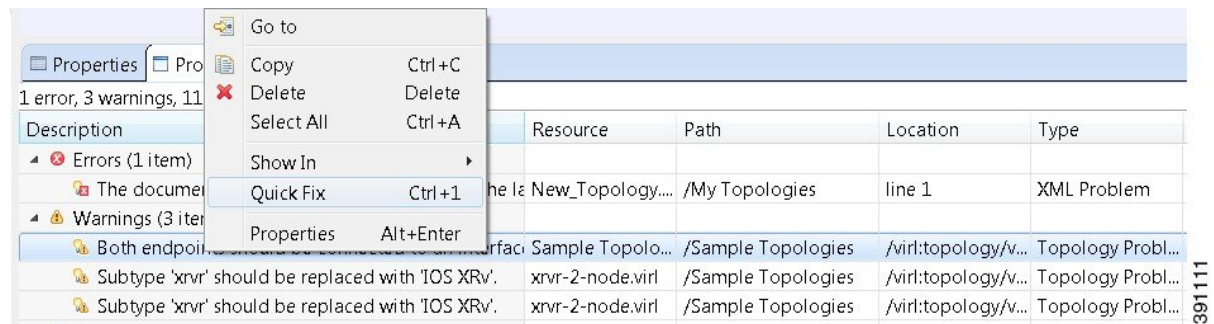
Icon	Function	Description
	View Menu	<p>The View menu has the following options:</p> <ul style="list-style-type: none"> • Show: Displays errors and warnings. • Group By: Groups problems under the headings Type, Severity, and None. The default is Severity. • Sort By: Sorts problems under the following headings: <ul style="list-style-type: none"> ◦ Description: Details the problem encountered. ◦ Resource: Displays the name of the .virl file where the problem has occurred. ◦ Path: Displays the applicable project folder. ◦ Location: Displays the location in the .virl file where the problem has occurred. ◦ Type: Displays the type of problem, for example, XML problem. • New Problems View: Opens a new Problems view on the workbench. • Configure Contents: Opens the Configure Contents dialog box, where you can add multiple filters to the Problems view and enable or disable them. Filters can either be additive or exclusive. The All Errors/Warnings on Selection filter is provided by default. • Configure Columns: Opens the Configure Columns dialog box, where you can choose to hide or show specific information about the problem encountered, as shown in the Sort By option. Options are: <ul style="list-style-type: none"> ◦ 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 25: Quick Fix

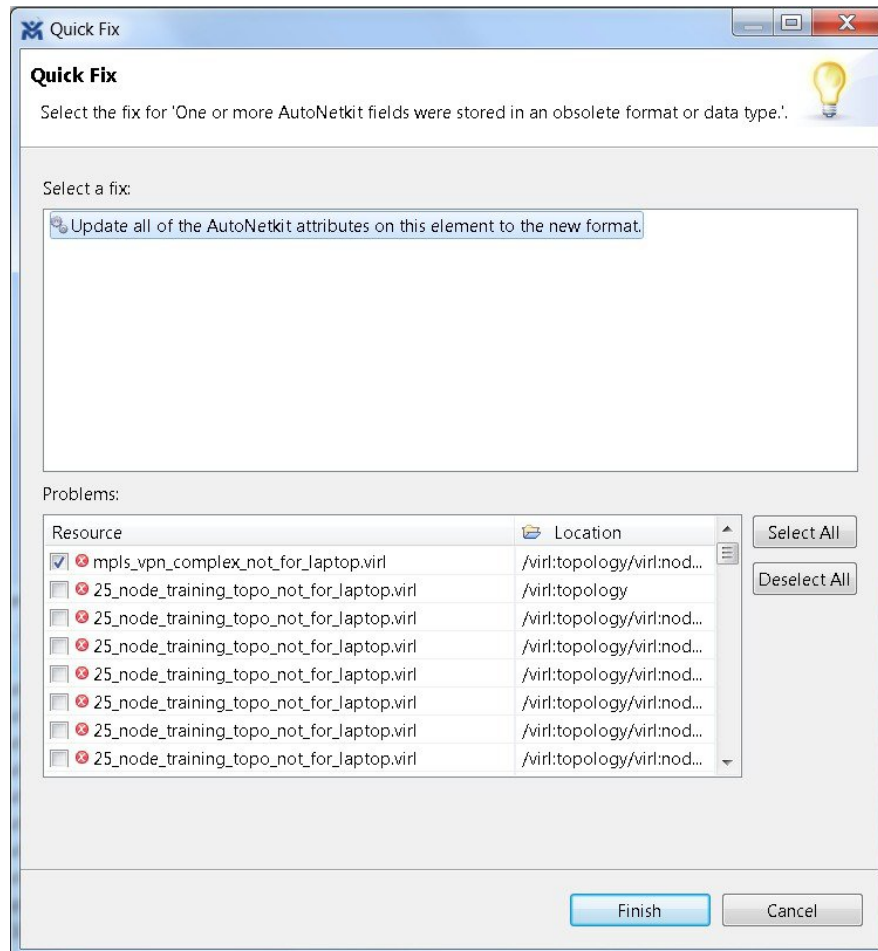


To fix a problem:

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

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

Figure 26: Quick Fix Dialog Box



- 2 Select a fix from the list, and then check the check box of any of the resources listed in the **Problems** area. You can click **Select All** to apply the quick fix to all the resources listed. Alternatively, you can click **Deselect All** to clear all selections.
- 3 Click **Finish**.



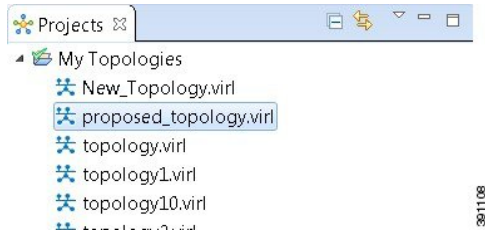
Note

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

Projects View

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

Figure 27: Projects View







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


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

Figure 28: Projects View Toolbar






Table 17: Available Tools

Icon	Function	Description
	Collapse All	Collapses the hierarchy of all the resources in the Projects view.
	Link with Editor	Links the Projects view with an active editor. A change to an active editor automatically updates the Projects view and allows you to toggle between the two views.
	View Menu	Provides options for customizing the content displayed in the Projects view. Options are: <ul style="list-style-type: none"> • 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:

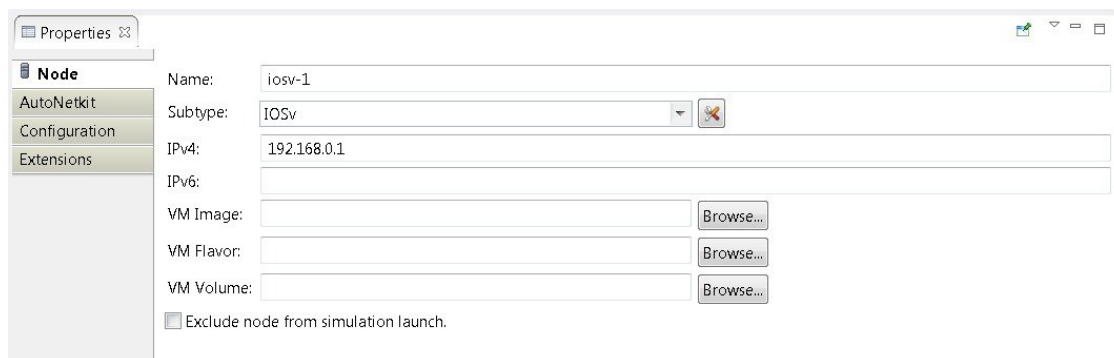
Table 18: Projects View Icons

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

Properties View

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

Figure 29: Properties View










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

Figure 30: Properties View Toolbar



Table 19: Available Tools

Icon	Function	Description
	Show Categories	Shows the available properties categories.
	Show Advanced Properties	Shows all advanced properties.
	Restore Default Value	Restores the default value for the property.
	Pin to Selection	Pins this properties view to the current selection.
	View Menu	Displays menu items that allow you to: <ul style="list-style-type: none"> • Open a new properties view. • Pin the properties view to the current selection.
	Minimize	Reduces the size of the Properties view.
	Maximize	Increases the size of the Properties view.

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

- [Node Properties](#), on page 41
- [Topology Properties](#), on page 46

Node Properties

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

Figure 31: Node Properties

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

Table 20: Node Properties

Property	Description
Name	Specify a name for the node. Note Use only alphanumeric and special characters for node names. Node names must be unique across the entire topology. Duplicate node names cause the build to fail when the configuration is autogenerated. If a duplicate node name is defined, a marker is shown in the Problems view. Unicode is not supported for node names. The use of a period in a node name may cause the node name to wrap when viewing the hierarchy from the Cisco Modeling Labs server. When you update a node name in the Properties > Node view, the node name is not updated in the Node Editor .
Subtype	Specify a subtype from the list. Cisco Modeling Labs 1.1 includes the following images built into the Cisco Modeling Labs client: <ul style="list-style-type: none"> • Cisco IOSv Software Release 15.4(3)M • Cisco IOSv Layer 2 switch • Ubuntu 14.04 Cloud Additional Cisco virtual images are available for use. However, they must be installed separately. For a list of supported subtypes, see Release Notes for Cisco Modeling Labs 1.1 . Note When changing a node subtype after it is initially configured using AutoNetkit, for example, changing a Cisco IOSv subtype to a Cisco IOS XRv subtype, you need to revalidate the AutoNetkit properties for the new node subtype.

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

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

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

**Note**

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

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

Table 21: AutoNetkit Properties

Property	Fields	Description
General	ASN	Specify the autonomous system number, which is used to infer IGP and BGP. This can be any valid integer.
IGP	IGP	Configure an internal routing protocol. Options are: <ul style="list-style-type: none"> • Not specified • OSPF • ISIS • EIGRP • RIP-V2 <p>The default value is Not specified.</p>
	OSPF Area	Configure an OSPF area. The default value is 0 .

Property	Fields	Description
iBGP	iBGP Role	Configure an iBGP role from the list and use it to create an iBGP topology. Options are: <ul style="list-style-type: none"> • Not specified • Disabled • Peer • RR (route reflector) • HRR(hierarchical route reflector) • RRC(route reflector client) <p>The default value is Peer.</p>
	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.
Custom Configuration	Global	In this section, users can specify their own configuration text for inclusion in the appropriate section of the node configuration. Note The following fields do not apply to external routers. Note too that text entered is not syntactically checked, so ensure that the text is valid.
	Physical Interfaces	Specify a custom configuration for the physical interfaces.
	Loopback Zero	Specify a custom configuration for loopback zero.
	OSPF	Specify a custom configuration for OSPF.
	IS-IS	Specify a custom configuration for IS-IS.
	EIGRP	Specify a custom configuration for EIGRP.
	RIP-V2	Specify a custom configuration for RIPv2.
	BGP	Specify a custom configuration for BGP.
MPLS	VRF Name	Specify an MPLS VPN name for VRF MPLS VPNs.
	LDP	Enable Cisco MPLS Label Distribution Protocol. Options are: <ul style="list-style-type: none"> • Not specified • True • False <p>The default value is False.</p>

Property	Fields	Description
	Enable MPLS TE	<p>Enable Cisco MPLS Traffic Engineering. Options are:</p> <ul style="list-style-type: none"> • Not specified • True • False <p>The default value is False.</p>
External BGP	IPv4 Address	<p>Note This property only applies to external routers.</p> <p>Specify the IPv4 address of the remote router. Enter a valid IP address in the correct format.</p>
	IPv6 Address	Specify an IPv6 address of the remote router. Enter a valid IP address in the correct format.
	Remote ASN	Specify the AS number of the remote router. This is used when trying to establish a BGP connection to a remote device. The value range is 1 to 65535.
	MD5 Password	Specify the MD5 password to use to secure the BGP session to the remote router.
	Multihop	<p>Enable BGP multihop. When the remote router is directly adjacent (Layer 3 adjacent), this field is set to False; otherwise it is set to True.</p> <p>Options are:</p> <ul style="list-style-type: none"> • Not specified • True • False <p>The default value is True.</p>
External L2TPv3	Remote Loopback IPv4 Address	Specify a remote loopback IPv4 address.
	Local Endpoint IPv4 Address	Specify a local endpoint IPv4 address.
	Local Endpoint IPv4 Netmask	Specify a local endpoint IPv4 netmask.
	PseudoWire ID	Specify a pseudowire ID.

Property	Fields	Description
GRE Tunnel	IPv4 Tunnel Enabled	Enable IPv4 GRE tunneling. Options are: <ul style="list-style-type: none"> • Not specified • True • False The default value is False .
	Tunnel IPv4 Address	Specify a tunnel IPv4 address to use, which is the IP address of the far-end node terminating the GRE tunnel itself.
	Tunnel IPv4 Netmask	Specify a tunnel IPv4 netmask to use.
	IPv6 Tunnel Enabled	Enable IPv6 GRE tunneling. Options are: <ul style="list-style-type: none"> • Not specified • True • False The default value is False .
	Tunnel IPv6 Address	Specify a tunnel IPv6 address to use, which is the IP address of the far-end node terminating the GRE tunnel itself.
	Tunnel IPv6 Netmask	Specify a tunnel IPv6 netmask to use.

For this release, RIP-V2 has been added as an IGP in addition to IS-IS, OSPF, and EIGRP. You can specify the IGP on a per-node basis, which takes precedence over the global IGP setting. Additionally, you can specify LDP enabled/disabled per node, which allows LDP to be enabled directly, rather than only when VRFs are specified.

Per interface you can specify a VLAN. This is specified on the destination interface and determines the VLAN a node belongs to. This is used in the Cisco IOSvL2 configuration. See [Assign VLANs, on page 104](#) for more information

Information displayed under the **Configuration** tab depends on whether the **Auto-generate the configuration based on these attributes** check box under the **AutoNetkit** tab is checked.

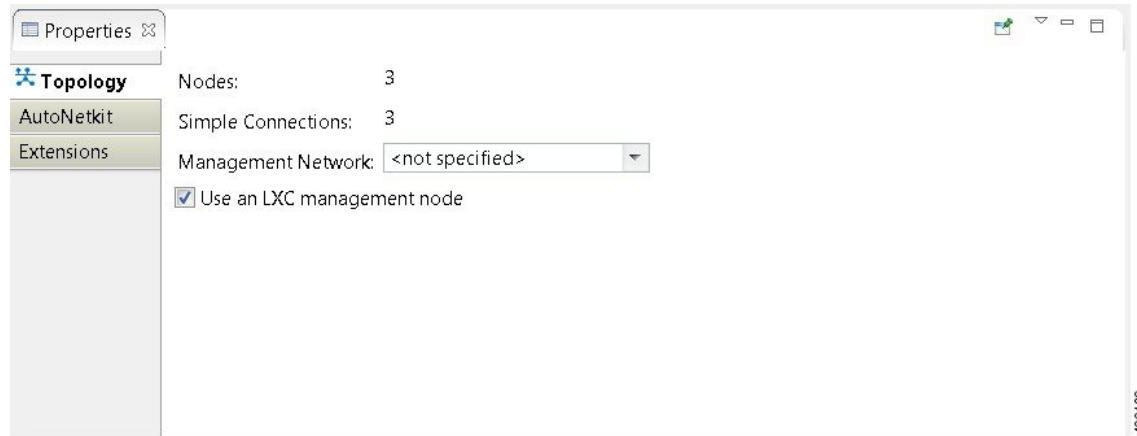
- When checked, AutoNetkit generates the configuration and displays it under the node's **Configuration** tab.
- When unchecked, no configuration information is created for the node. You must configure the node manually or cut and paste the existing configuration information into this area.

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

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 32: Topology Properties



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

Table 22: Topology Properties

Property	Description
Management Network	Specify the type of out-of-band (OOB) external network access. Options are: <ul style="list-style-type: none"> • Not specified • Private simulation network: Creates a per-simulation Linux container (LXC). The LXC is automatically connected into the OOB management network to which all VMs in your simulation are connected, enabling you to connect into each VM via its management Ethernet port. This removes the need to use the console port connection method. See the section Linux Container (LXC), on page 137 for more information. • Shared flat network: Enables OOB external network access to all devices in the topology. • Private project network: Enables OOB private access to all simulations running within the user space.

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



Note

In this instance, **Nodes** include node subtypes and FLAT and SNAT port groups.

The **Use an LXC management node** check box is used to enable the management node as a Linux container. See the section [Linux Container \(LXC\)](#), on page 137 for more information.

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

Table 23: AutoNetkit Properties

Property	Fields	Description
General	Enable CDP	Enable the Cisco Discovery Protocol (CDP). Options are: <ul style="list-style-type: none"> • Not specified • True • False The default value is False .
	Enable OnePK	Enable the Cisco One Platform Kit (OnePK). Options are: <ul style="list-style-type: none"> • Not specified • True • False The default value is False .
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: <ul style="list-style-type: none"> • 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 .
	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 Prefix	Specify the prefix size to use for IPv4 loopback address allocations. The default value is 22 .

Property	Fields	Description
	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 .
	IPv6 Infrastructure Subnet	Specify the address to use for IPv6 infrastructure address allocations. This is the address assigned to the interface created on the router. The default value is 0:0:0:a:: . Enter a valid IP address in the correct format.
	IPv6 Infrastructure Prefix	Specify the prefix to use for IPv6 infrastructure address allocations. The default value is 64 .
	IPv6 Loopback Subnet	Specify the address to use for IPv6 loopback address allocations. The default value is 0:0:0:b:: . Enter a valid IP address in the correct format.
	IPv6 Loopback Pool Prefix	Specify the prefix size to use for IPv6 loopback address allocations. The default value is 64 .
	IPv6 VRF Subnet	Specify the address to use for IPv6 VRF address allocations when specifying the address range for MPLS VRF. The default value is 0:0:0:c:: . Enter a valid IP address in the correct format.
	IPv6 VRF Prefix	Specify the prefix to use for IPv6 VRF address allocations. The default value is 64 .
Routing	Enable Routing Protocols	<p>Configure routing protocols (BGP and IGP). Options are:</p> <ul style="list-style-type: none"> • Not specified • True • False <p>The default value is True. If you specify False, there will be no router configuration for any of the routing protocols.</p>

Property	Fields	Description
	IGP	Configure the Cisco IGP. Options are: <ul style="list-style-type: none"> • Not specified • OSPF • ISIS • EIGRP • RIP-V2 The default value is OSPF .
MPLS	Enable MPLS OAM	Enable Cisco MPLS OAM for all routes on the topology. Options are: <ul style="list-style-type: none"> • 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 24: IP Address Default Values

Viewed from	Option	Default Value	Optional Value(s)
Topology > AutoNetkit > Addressing	IP Address Family	v4	Not specified, None, v6, dual_stack
	IPv4 Infrastructure Subnet	10.0.0.0	Address to use.
	IPv4 Infrastructure Prefix	8	Prefix to use.
	IPv4 Loopback Subnet	192.168.0.0	Address to use.
	IPv4 Loopback Pool Prefix	22	Prefix to use.
	IPv4 VRF Subnet	172.16.0.0	Address to use.
	IPv4 VRF Prefix	24	Prefix to use.
	IPv6 Infrastructure Subnet	0:0:0:a::	Address to use.
	IPv6 Infrastructure Prefix	64	Prefix to use.
	IPv6 Loopback Subnet	0:0:0:b::	Address to use.

Viewed from	Option	Default Value	Optional Value(s)
	IPv6 Loopback Pool Prefix	64	Prefix to use.
	IPv6 VRF Subnet	0:0:0:c::	Address to use.
	IPv6 VRF Prefix	64	Prefix to use.

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

Table 25: Routing Protocols Default Values

Viewed from	Option	Default Value	Optional Value(s)
Topology > AutoNetkit > General	Enable CDP	false	true, Not specified
	Enable OnePK	false	true, Not specified
Topology > AutoNetkit > Routing	Enable Routing Protocols	true	false, Not specified
	IGP	OSPF	ISIS, EIGRP, RIPv2, Not specified
Topology > AutoNetkit > MPLS	Enable MPLS OAM	false	true, Not specified
Node > AutoNetkit > General	ASN	1	None or any valid integer
Node > AutoNetkit > IGP	IGP	OSPF	ISIS, EIGRP, RIPv2, Not specified
	OSPF Area	0	None or valid OSPF area number
Node > AutoNetkit > iBGP	iBGP Role	Peer	Disabled, RRC, HRR, RR, Not specified
	RR Cluster	No default value	None or alphanumeric string
	HRR Cluster	No default value	None or alphanumeric string
Node > AutoNetkit > Custom Configuration	Global	No default value	Specify a custom configuration for the global stanza.

Viewed from	Option	Default Value	Optional Value(s)
	Physical Interfaces	No default value	Specify a custom configuration for the physical interfaces.
	Loopback Zero	No default value	Specify a custom configuration for loopback zero.
	OSPF	No default value	Specify a custom configuration for OSPF.
	IS-IS	No default value	Specify a custom configuration for IS-IS.
	EIGRP	No default value	Specify a custom configuration for EIGRP.
	RIP-V2	No default value	Specify a custom configuration for RIP-V2.
	BGP	No default value	Specify a custom configuration for BGP.
Node > AutoNetkit > MPLS	VRF Name	No default value	None or alphanumeric string
	Enable MPLS TE	false	true, Not specified
Node > AutoNetkit > External BGP	Multihop	true	false, Not specified
Node > AutoNetkit > GRE Tunnel	IPv4 Tunnel Enabled	false	true, Not specified
	IPv6 Tunnel Enabled	false	true, Not specified

Under the **Extensions** tab, all the extensions used to generate the configuration are listed 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 26: Interface Properties

Property	Description
Name	Specify a name for the interface.
IPv4	Specify an IPv4 interface address and an IPv4 subnet prefix length.
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 with the **Key**, **Value**, and **Type** attributes.

Connection Properties

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

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

Search View

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

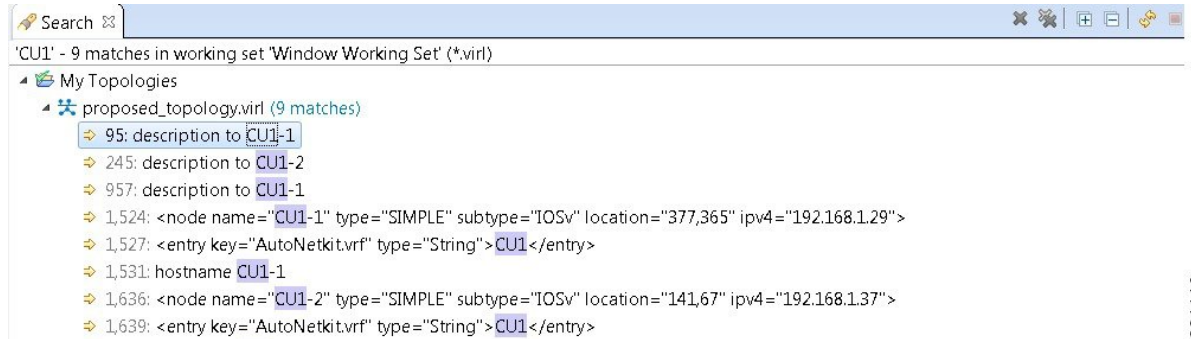
**Note**

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

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

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

Figure 33: Search View



391116

To use the search functionality, perform the following tasks:












- 1 From the Cisco Modeling Labs client toolbar, click **Search > File**. The **Search** dialog box appears.
- 2 In the **Containing text** field, enter the text string to search for. The **Containing text** field displays a list of recently performed searches to select from. Leave this field empty if you want to search for files only. Check or uncheck the **Case sensitive** check box depending on whether a case-sensitive search is to be performed. You can also check the **Regular expression** check box to enable more powerful searching capabilities. Check the **Whole word** check box if you want to search for whole words that are identical to the text string. Specify the types of files to include in the search in the **File name patterns** field.
- 3 Click **Choose** to open the **Select Types** dialog box. This dialog box provides a quick way to select from a list of valid extensions.
- 4 In the **Scope** area, specify the files and folders to include in the search. Valid options are:
 - The entire workspace.
 - The currently selected resources in the workspace.
 - A named working set.
 - A customized group of files and folders. Use the **Customize** option to define the type of available searches from the **Search Page Selection** dialog box.
- 5 Click **Search** to begin your search. The **Search** view appears with the results of the search listed. You can click the **Cancel** tool in the **Search** view to cancel your search while it is still in progress.

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

Figure 34: Search View Toolbar



Table 27: Available Tools

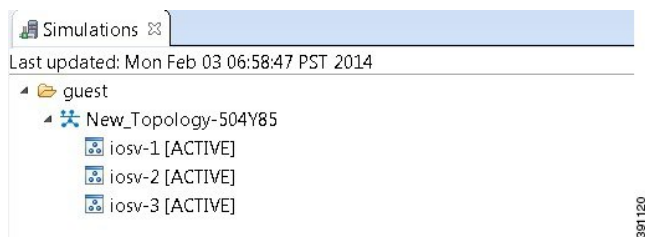
Icon	Function	Description
	Remove Selected Matches	Deletes all the highlighted matches from the search results.
	Remove All Matches	Deletes all the matches from the search results.
	Expand All	Expands each item in the Search view.
	Collapse All	Collapses each item in the Search view.
	Run the Current Search Again	Reruns the current search to retrieve previous search results or to reflect recent changes.
	Cancel Current Search	Cancels the search currently running.
	Show Previous Searches	Browses previously conducted searches and selects a previous search from the drop-down menu to repeat a previous search. You can also clear the search history.
	Pin the Search View	Pins the Search view so that subsequent search results are displayed in a separate Search view while the pinned view remains unchanged. This allows for a comparison of results.
	View Menu	Displays the search results as a tree or a list, filters the results using the Filters option, and sets the overall preferences for searches using the Preferences option.
	Minimize	Reduces the size of the Search view.
	Maximize	Increases the size of the Search view.

Simulations View

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

- Name of the user running the simulation
- Name of the topology
- Number of nodes in the running simulation
- Current state of each node

Figure 35: Simulations View



Possible simulation states are:

Table 28: 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.
INIT	Indicates that the simulation has just been created and is being launched.
STOP	Indicates that a stop simulation request has been received.

Possible node states are:

Table 29: Node States

State	Description
ACTIVE	Indicates that the VM process is successful. It may take a few minutes for the node to boot up and configure.
ABSENT	Indicates that the VM is not currently deployed.
BUILDING	Indicates that the VM is starting but the router image has not yet loaded.

State	Description
ERROR	Indicates that the VM process failed.
PAUSED and SUSPENDED	<p>These two states can occur when:</p> <ul style="list-style-type: none"> • The host environment is rebooted. • The node itself shuts down. • The operating system of the node crashes. • The virtualization process running the node crashes. <p>In such instances, the appropriate action is to stop the nodes and then start the nodes again.</p> <p>Note All configuration changes to the nodes are lost.</p>
TERMINATING	Indicates that a request to stop the VM process has been received, without regard to the current state of the VM.
SHUTOFF	<p>Indicates that the VM is deployed but not actually running. This state can occur when:</p> <ul style="list-style-type: none"> • The host environment is rebooted. • The node itself shuts down. • The operating system of the node crashes. • The virtualization process running the node crashes. <p>In such instances, the appropriate action is to stop the nodes and then start the nodes again.</p> <p>Note All configuration changes to the nodes are lost.</p>
UNKNOWN	Indicates that OpenStack is unresponsive and retrieval of node states' fails.



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

Figure 36: Simulations View Toolbar



Table 30: 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 37: Topology Options

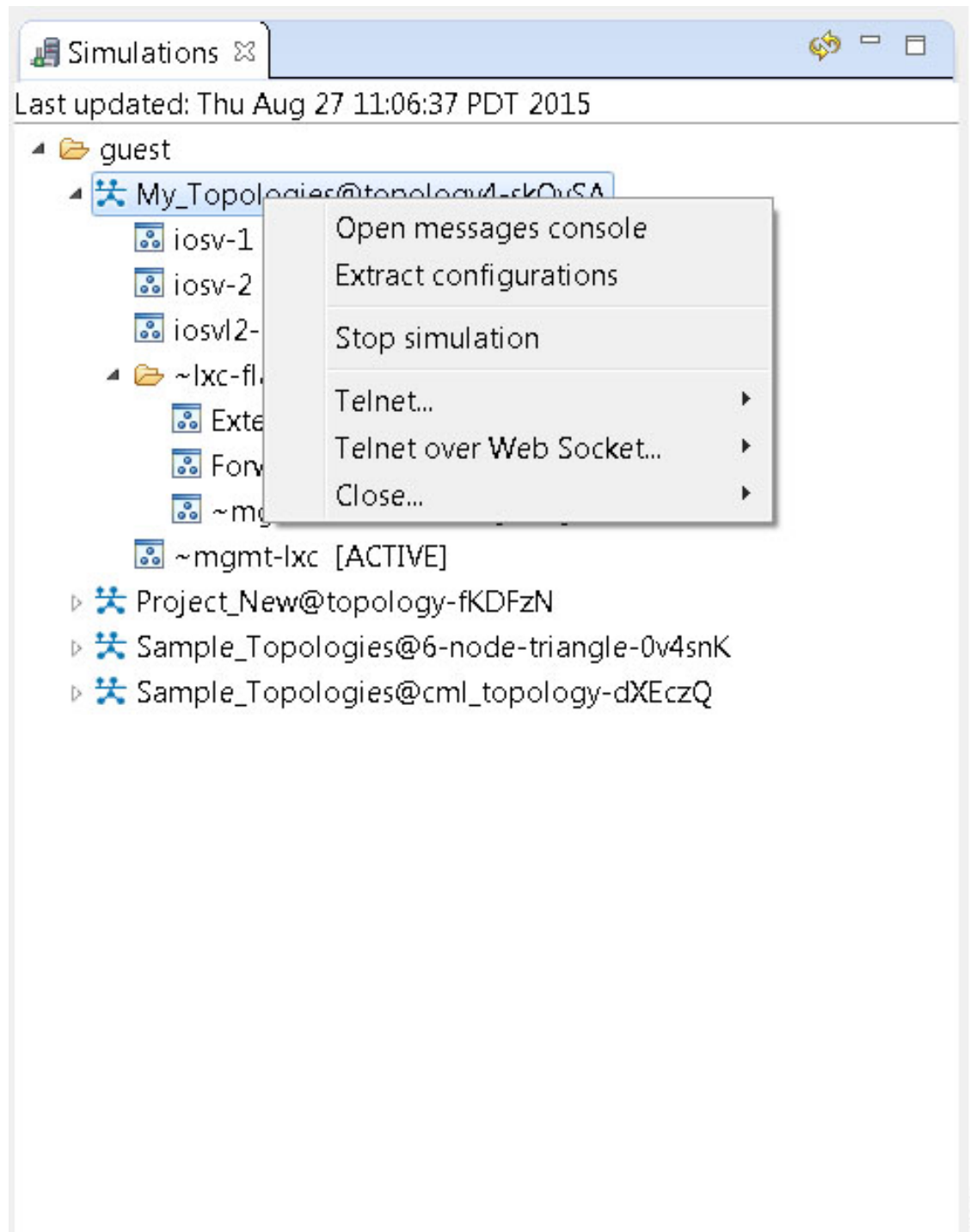


Table 31: Topology Options

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 all the routers' configurations to a locally saved file. 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 Simulate the Topology Overview, on page 131 for more information on stopping and starting simulations.
Telnet	Allows you to use Telnet to connect to ports on a node.
Telnet over WebSocket	Allows you to use Telnet to connect over WebSockets to ports on a node. WebSockets provide full-duplex communications channels over a single connection.
Close	Closes all port connections.

Node Options

The following operations are available when you right-click the node name in the **Simulations** view:

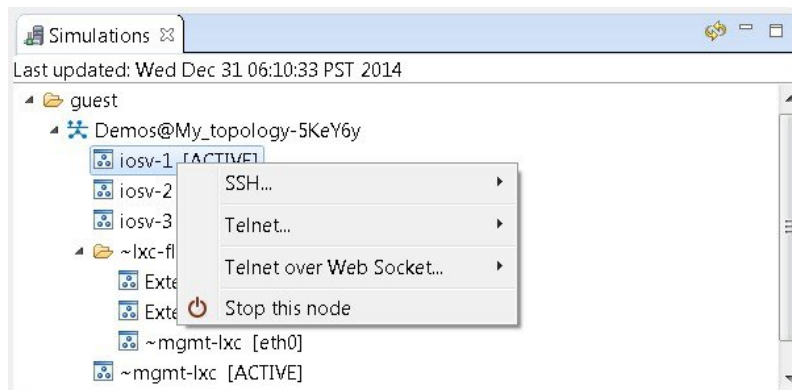
Figure 38: Node Options

Table 32: Node Options

Operation	Description
SSH	Allows you to use SSH to connect to ports on a node; this is available when the Management Network option Private Simulation Network is selected for the topology.
Telnet	Allows you to use Telnet to connect to ports on a node.
Telnet over Web Socket	Allows you to use Telnet to connect over WebSocket to ports on a node. WebSocket provides full-duplex communications channels over a single connection.
Stop this Node	Stops the selected node.
Start this Node	Starts the selected node.

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 39: Terminal View

```

iosv-1 (Console) - guestNew_Topology-i65u4s.virt.iosv-1 | iosv-2 (Console) - guestNew_Topology-i65u4s.virt.iosv-2
Telnet: (192.168.32.131:17016 - CONNECTED)
name Router


*Jan 30 18:24:53.846: %PLATFORM-5-SIGNATURE_VERIFIED: Image 'flash0:/vios-adventerprisek9-m' passed code signing verification
*****
* IOSv - Cisco Systems Confidential
*
* This software is provided as is without warranty for internal
* development and testing purposes only under the terms of the Cisco
* Early Field Trial agreement. Under no circumstances may this software
* be used for production purposes or deployed in a production
* environment.
*
* By using the software, you agree to abide by the terms and conditions
* of the Cisco Early Field Trial Agreement as well as the terms and
* conditions of the Cisco End User License Agreement at
* http://www.cisco.com/go/eula
*
* Unauthorized use or distribution of this software is expressly
* Prohibited.
*****





```

391125

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

Figure 40: Terminal View Toolbar**Table 33: Available Tools**

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

Icon	Function	Description
	Scroll Lock	Sets scrolling on and off.
	Display Selected Connections	Allows you to select a connection from the list of active terminal connections.
	Remove Terminal	Closes the Terminal view.
	Set Terminal Font	Allows you to set the font to be used in the terminal, from the Colors and Fonts dialog box. Note You can also access the Colors and Fonts dialog box by choosing File > Preferences > General > Appearance > Colors and Fonts .

Setting Preferences for the Cisco Modeling Labs Client

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

- [Node Subtypes Setting](#), on page 62
- [Terminal Setting](#), on page 64
- [Topology Editor Setting](#), on page 69
- [Web Services Setting](#), on page 70
- [AutoNetkit Visualization Setting](#), on page 72
- [Web Browser Setting](#), on page 75
- [Secure Storage Setting](#), on page 76

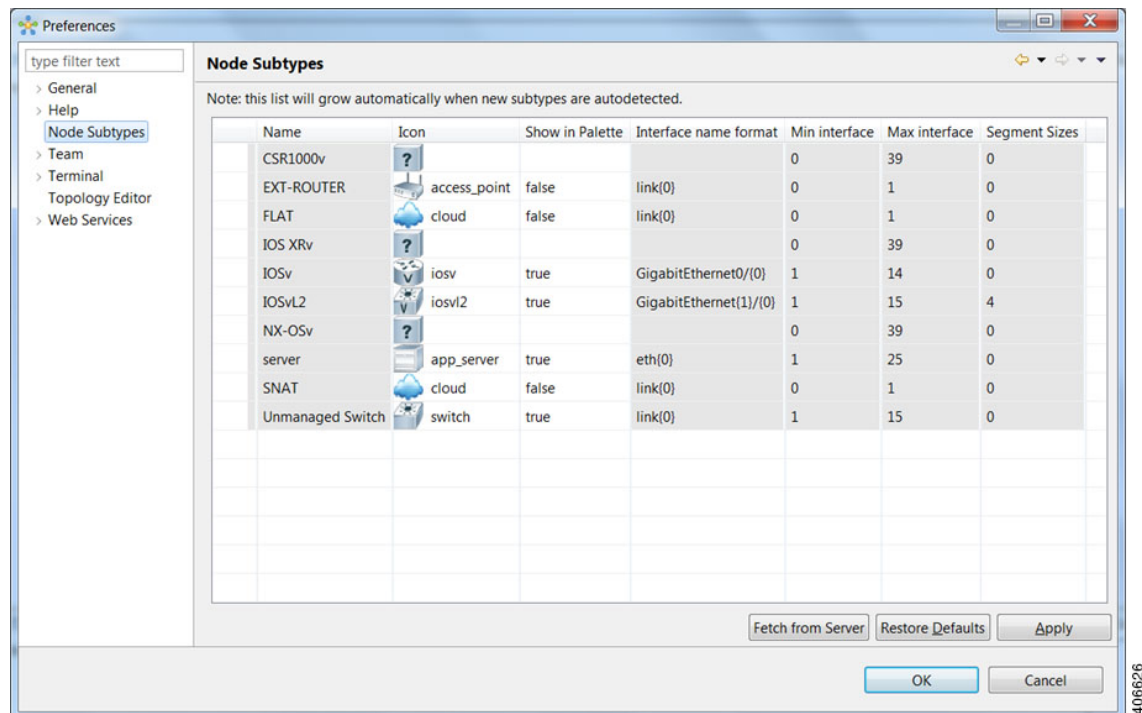
These are discussed in the following sections.

Node Subtypes Setting

A node subtype defines a VM running a particular operating system. For example, Cisco IOSv, Cisco IOSv Layer 2 switch software releases, and an Ubuntu 14.4.2 LTS Cloud-init image are installed with Cisco Modeling Labs 1.1. Additionally, Cisco IOS XRv Software Release 5.1.3 and Cisco CSR1000v Software Release 3.1.6 demonstration images are included in the build. Cisco ASAv Software Release 9.3.2 production image is available to purchase separately.

This setting allows you to add and remove node subtypes for use with the Cisco Modeling Labs 1.1 client. See the *User Workspace Management* chapter in the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide, Release 1.1* for information on installing VM images on the Cisco Modeling Labs server.

Figure 41: Node Subtypes Setting



The available operations for this setting are:

Table 34: Node Subtypes Setting Operations

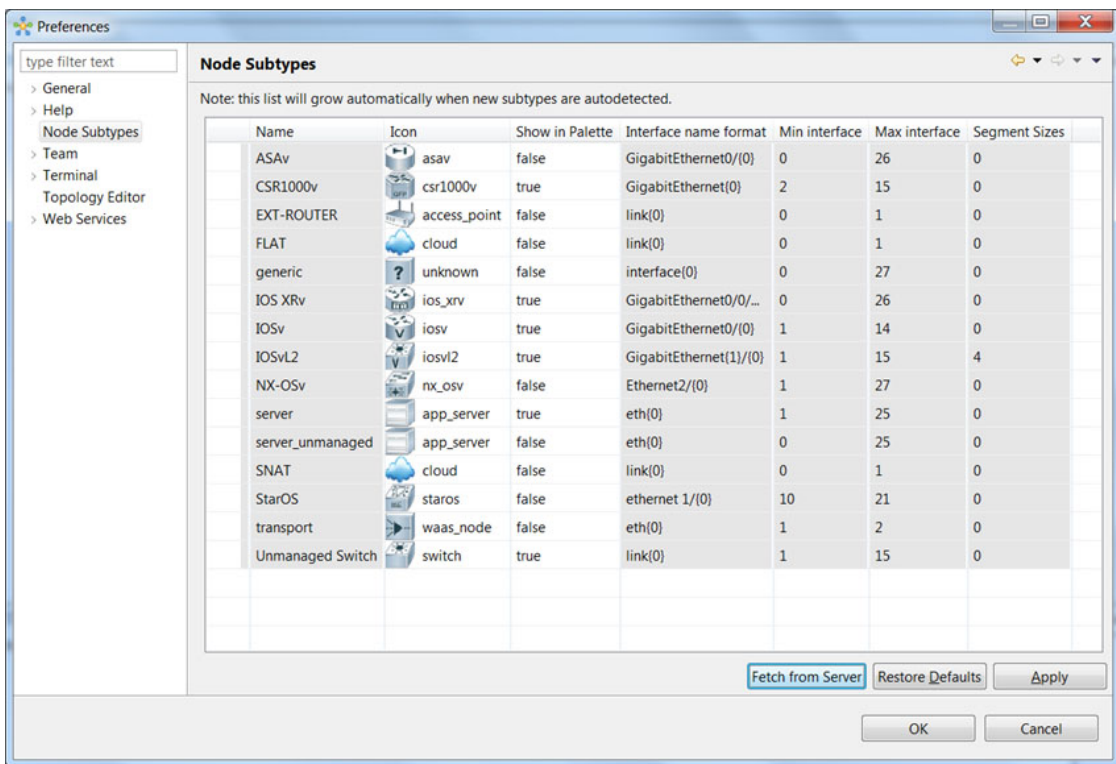
Operation	Description
Fetch from Server	Updates the local subtypes based on the currently configured Cisco Modeling Labs server. All the subtypes supported on Cisco Modeling Labs server are available with this operation. See Fetch Node Subtypes from the Cisco Modeling Labs Server, on page 64 .
Restore Defaults	Reverts to the original list of subtypes.
Apply	Applies changes.

Fetch Node Subtypes from the Cisco Modeling Labs Server

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

- Step 1** Click **File > Preferences > Node Subtypes**.
- Step 2** Click the **Fetch from Server** button.
The **Confirm** dialog box is displayed.
- Step 3** Click **OK** to update the list of node subtypes.

Figure 42: Fetch Node Subtypes from Server



- Step 4** Click **OK** to finish.

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

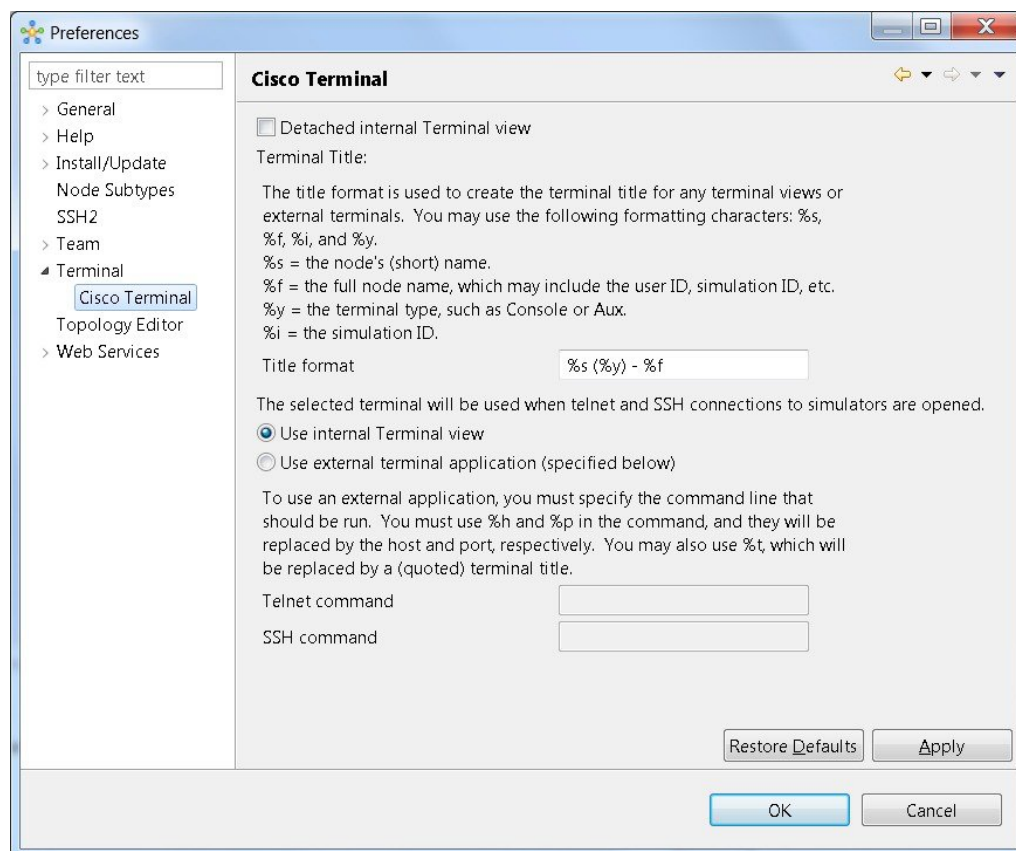
Contact your system administrator if a specific node subtype is missing from the list, as the system administrator is responsible for adding new node subtypes to the Cisco Modeling Labs server.

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** and **Simulation** perspectives. However, a detached view is only visible from the perspective in which it was detached.
- If you are using an external terminal application, you must specify both Telnet and SSH run commands. You must also ensure that the title format includes the percentage (%) character. Omission of either of these requirements will impede your ability to save your setting preferences.
- When you specify to use an external terminal via **File > Preferences > Terminal > Cisco Terminal** and then Telnet over WebSocket to a VM, the terminal opens internally, not externally as specified.

Figure 43: Terminal Setting

The available operations for this setting are:

Table 35: Terminal Setting Operations

Operation	Description
Detached Internal Terminal View	Launches the Terminal view in a separate window.

Operation	Description
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.
Use External Terminal Application (specified below)	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.

Setting Up an External Terminal

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

Windows

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



Note

The complete PATH must be specified.

Table 36: PuTTY Command-Line Options

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

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

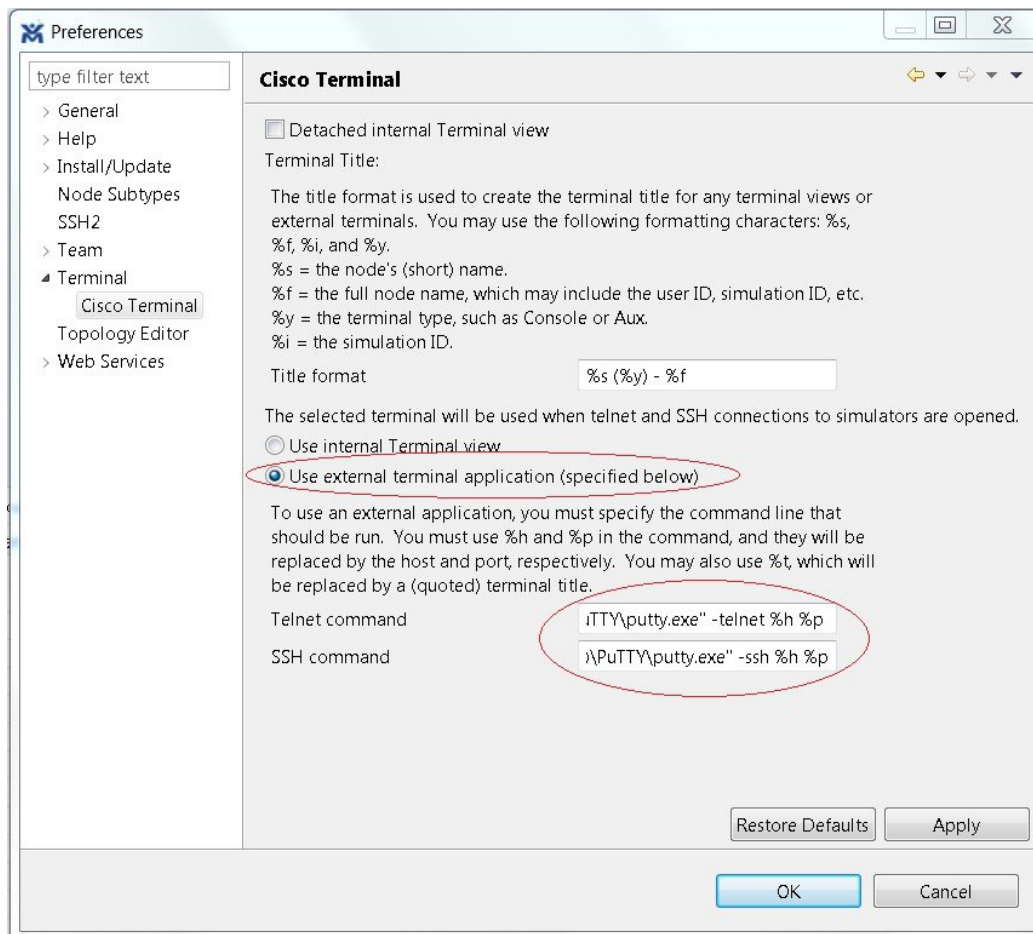
- Telnet command: `"C:\Program Files (x86)\PuTTY\putty.exe" -telnet %h %p`

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

**Note**

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

Figure 44: Specify an External Terminal Application for Use



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

Table 37: Additional Terminal Programs

Terminal Program	Connection Type	Command to Use
Xshell	Telnet	"C:\Program Files (x86)\NetSarang\Xshell 4\xshell.exe" -url telnet://%h:%p -newtab %t

Terminal Program	Connection Type	Command to Use
SecureCRT	Telnet	"C:\Program Files\VanDyke Software\SecureCRT\SecureCRT.exe" /T /TELNET %h %p Note The /T option ensures SecureCRT creates a tab for new sessions instead of opening a new window. Ensure to validate the path of the binary.

OS X

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

- AppleScript: A program used to start applications and interact with them; some examples are open windows, start new sessions, paste keyboard input into a session, and so on.
- /usr/bin/osascript: A built-in OS X command-line utility used to execute AppleScript and other OSA language scripts. This is configured in the Cisco Modeling Labs client; it is essentially the glue between the Cisco Modeling Labs client and the terminal application.

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

Table 38: Additional Terminal Programs for OS X

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

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

iTerm

```
on run argv
  tell application "iTerm"
    activate
    if current terminal exists then
      set t to current terminal
    else
      set t to (make new terminal)
    end if

    tell t
      launch session "Default Session"
```

```

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

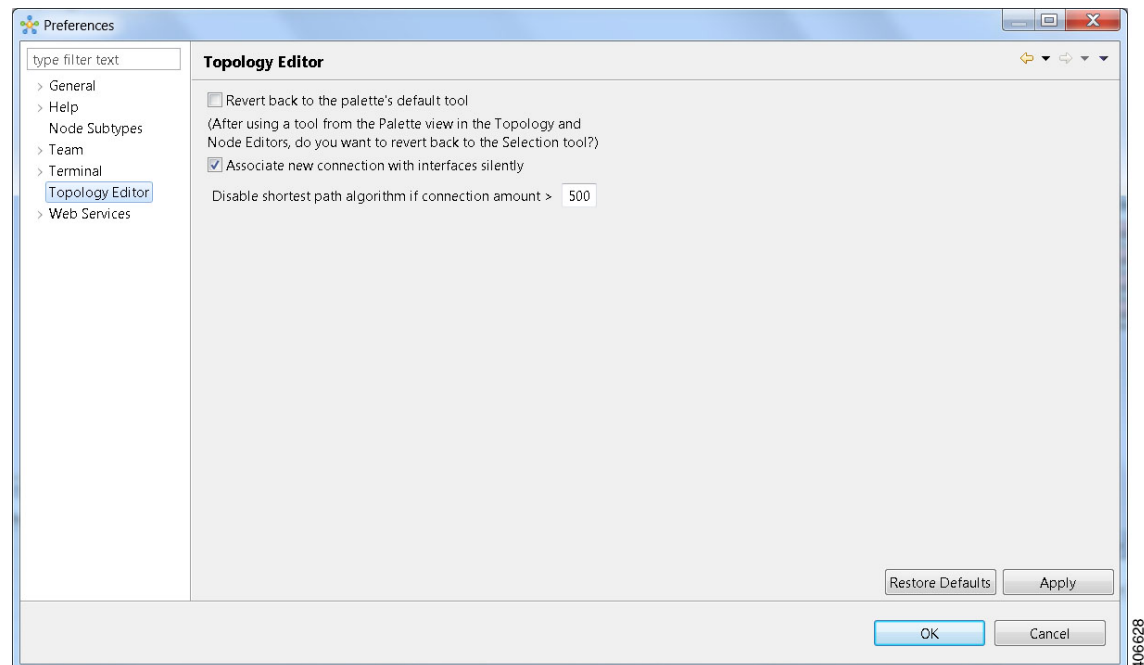
Terminal.app
on run argv
    tell application "Terminal"
        activate
        -- open a new Tab - there is no method
        tell application "System Events"
            keystroke "t" using {command down}
        end tell
        repeat with win in windows
            try
                if get frontmost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

```

Topology Editor Setting

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

Figure 45: Topology Editor Setting



The available operations for this setting are:

Table 39: Topology Editor Setting Operations

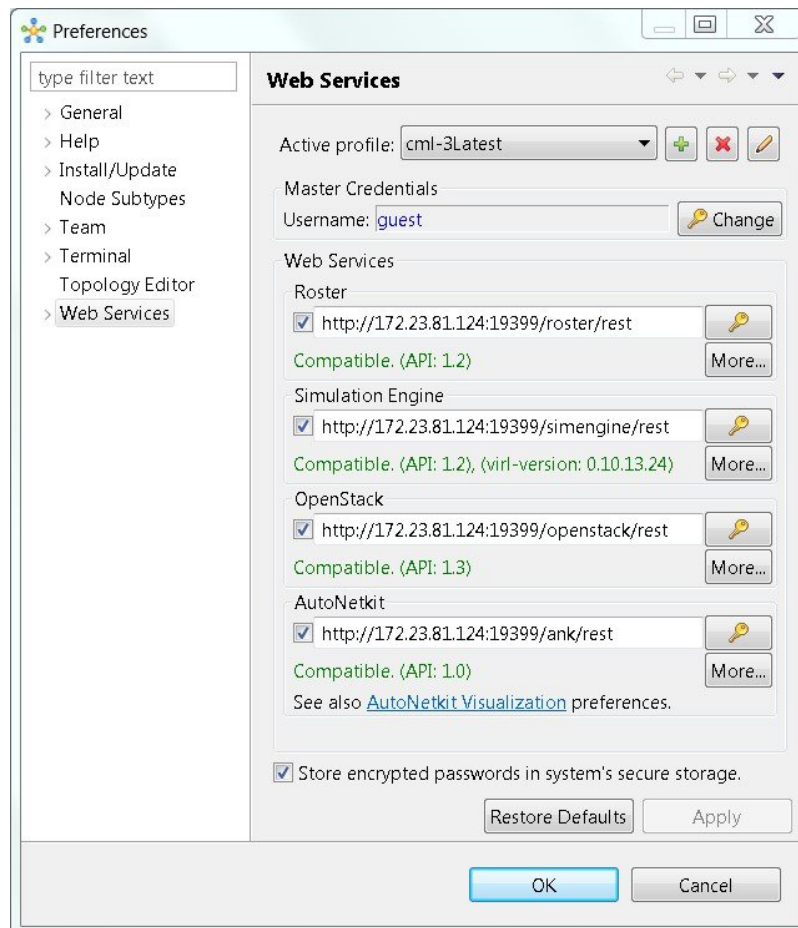
Operation	Description
Revert Back to the Palette's Default Tool	Resets the definition of the Palette view tools Select and Connect to their default configuration. This option is disabled by default.
Associate New Connection with Interfaces Silently	Enables the following: <ul style="list-style-type: none"> • 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: <ul style="list-style-type: none"> • 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 **Unauthorized** in red. This message relates to the **Master Credentials**

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

Figure 46: Web Services Setting



The available operations for this setting are:

Table 40: 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	<p>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:</p> <ul style="list-style-type: none"> • Roster • Simulation Engine • OpenStack • AutoNetkit <p>Note After you have set up your profile with the correct base URI, each of the web services will display Compatible in green, indicating that the Cisco Modeling Labs client can communicate with the Cisco Modeling Labs server.</p>
Store Encrypted Passwords in System's Secure Storage	<p>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 Secure Storage Setting, on page 76 for more information.</p>
Restore Defaults	Restores settings to the initial default state.
Apply	Applies changes.

AutoNetkit Visualization Setting

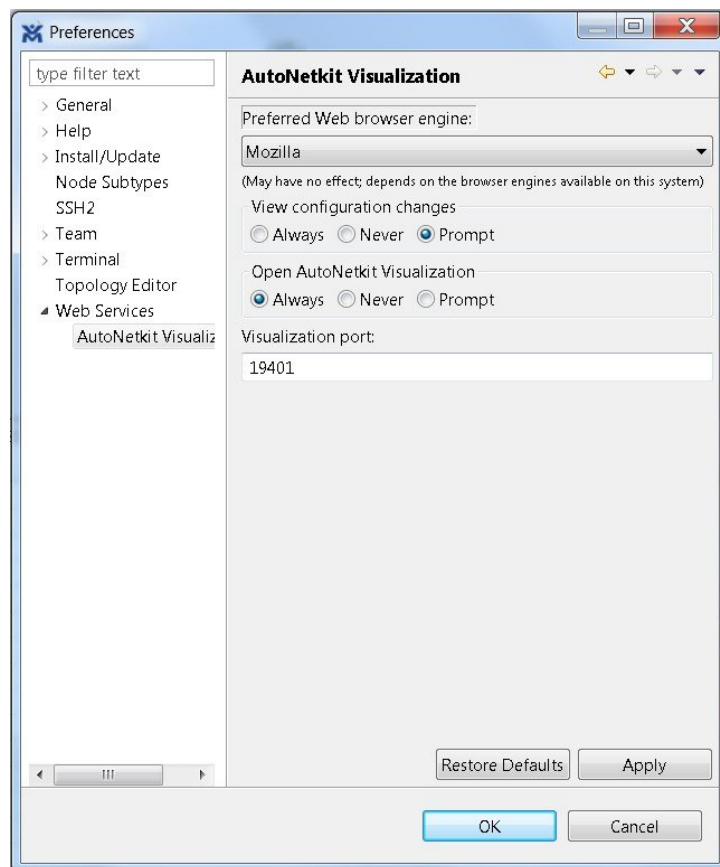
The AutoNetkit visualization feature is available only when node configurations are built using the parameters defined in AutoNetkit. If AutoNetkit visualization is turned off, you cannot get a true representation of your topology.

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 47: AutoNetkit Visualization Setting



The available operations for this setting are:

Table 41: AutoNetkit Visualization Setting Operations

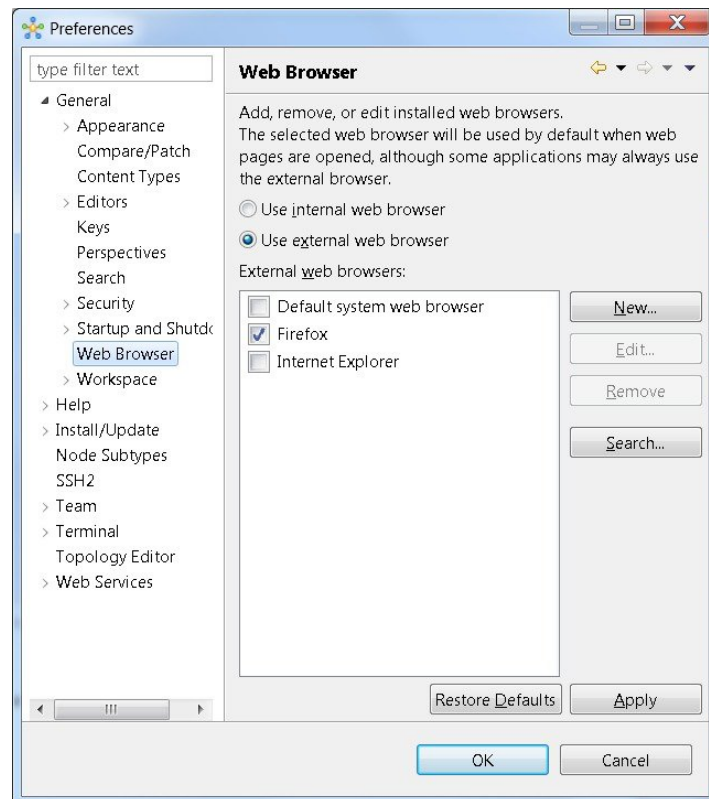
Operation	Description
Preferred Web Browser Engine	<p>Specifies the external web browser to use when displaying AutoNetkit visualization results. Options are:</p> <ul style="list-style-type: none"> • Default: The default system browser • WebKit: Based on the Safari web browser • Mozilla <p>Note You are 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.</p>

Operation	Description
View Configuration Changes	<p>Specifies whether to view configuration changes after a new build is generated. Options are:</p> <ul style="list-style-type: none"> • Always • Never • Prompt <p>The default value is Prompt.</p>
Open AutoNetkit Visualization	<p>Specifies when to open a browser window to display AutoNetkit visualization. Options are:</p> <ul style="list-style-type: none"> • Always • Never • Prompt <p>The default value is Prompt.</p>
Visualization Port	<p>Assigns a port value to the web service supporting AutoNetkit visualization. The default port is 19401. 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.</p> <p>Note If an incorrect port value is entered, you will be prompted with an error during the build stage.</p>
Restore Defaults	<p>Restores the default state for the preferred web browser engine and restores the default port value assigned to AutoNetkit visualization.</p>
Apply	<p>Applies changes.</p>

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 48: Web Browser Setting



The available operations for this setting are:

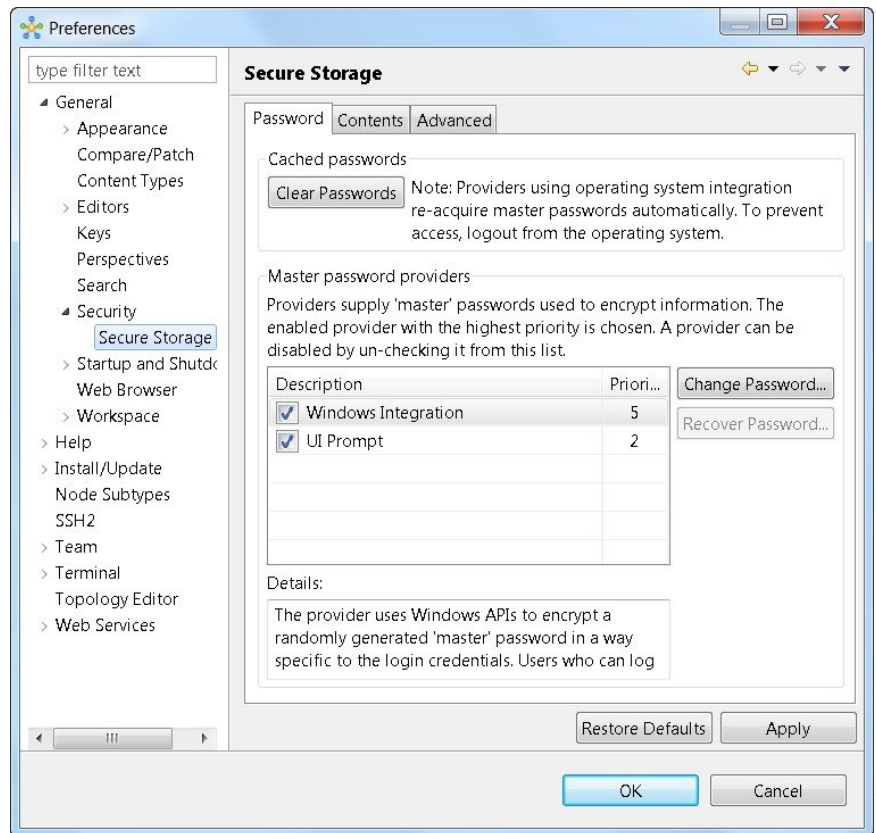
Table 42: Web Browser Setting Operations

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 49: Secure Storage Setting



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

The available options are:

Table 43: 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 recreate it to use the newly selected encryption algorithm.

Resetting the Secure Storage Password

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

Two methods are used to reset the password for the secure storage feature.

Method 1

- 1 From within Cisco Modeling Labs client, choose **File > Preferences > General > Security > Secure Storage**.
- 2 Click **Change Password**. The **Secure Storage** dialog box appears.
- 3 Click **Yes**. The **Password Recovery** dialog box appears.
- 4 Enter details in both Question fields and provide answers for both questions. Take note of the answers you provide, as these are treated as secondary passwords.
- 5 Click **OK**.

Method 2

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

- 1 Move to the `<user-home>/eclipse/org.eclipse.equinox.security` folder.
- 2 Delete the file `secure_storage`.
- 3 Open Cisco Modeling Labs client to provide details for the password for the secure storage feature when prompted.



Design a Topology

- [Design a Topology Overview, page 79](#)
- [Topology Nodes and Connections, page 79](#)
- [Create a Topology, page 82](#)
- [Place the Nodes on the Canvas, page 83](#)
- [Create Connections and Interfaces, page 84](#)
- [Use Unmanaged Switches, page 85](#)
- [The Cisco IOSvL2 Switch Image, page 86](#)
- [Create a Site, page 90](#)

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. See [Navigating Within the Cisco Modeling Labs Client, on page 7](#) for additional information about how to select and edit nodes and connection functions.

Topology Nodes

Table 44: Node Subtypes

Node Name	Node Type
Cisco IOSv	Router node. Runs a Cisco IOS operating system.
Cisco IOSvL2	Router node. Runs a Cisco IOS Layer 2 operating system.

Node Name	Node Type
Server	Server node. Runs a Linux operating system.
Cisco IOS XRv	Router node. Runs a Cisco IOS XR operating system.
Cisco CSR1000v	Router node. Runs a Cisco CSR 1000 operating system.
Cisco NX-OSv	Router node. Runs a Cisco NX-OS 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. See the *User Workspace Management* chapter in the *Cisco Modeling Labs Corporate Edition System Administrator Installation Guide, Release 1.1* 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 45: Node VM Images

VM Image Name	Used For
server	Server node
CSR1000v	Cisco CSR1000 node
IOSv	Cisco IOS node
IOSvL2	Cisco IOS Layer 2 node
IOS XRv	Cisco IOS XR node
NX-OSv	Cisco NX-OS node

Table 46: Node VM Flavors

VM Flavor Name	Used For
m1_tiny	Linux server
m1_small	Linux server
m1_medium	Linux server

VM Flavor Name	Used For
m1_large	Linux server
m1_xlarge	Linux server
server	Linux server
CSR1000v	Cisco CSR 1000 node
IOS XRv	Cisco IOS XR node
IOSv	Cisco IOS node
IOSvL2	Cisco IOS Layer 2 node
NX-OSv	Cisco NX-OS 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 47: 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.
External Router	Creates an external router connection point. When the external router is used in conjunction with a Layer 2 External (Flat) network and IOSv instances, AutoNetkit is able to configure an L2TPv3 tunnel to connect simulations to remote devices in a transparent manner.
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.

Connection Type	Description
Layer 2 External (Flat1)	Creates a Layer 2 external connection point using FLAT1. This second external connection point allows connections outside of Cisco Modeling Labs to connect to the topology.

Table 48: Site Groups

Type	Description
Site	Creates a site that uses levels of hierarchy in a Cisco Modeling Labs topology. Sites can be added to a topology, objects can be created within a site, and objects can be linked between sites and parent sites.

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 > Topology File**.
 - 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** 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** file is created in the selected project folder.
-

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 to place 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** (OS X) to toggle between node selections.

- From the menu bar, use **Edit > Distribute Nodes** 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 **Edit > Alignment** from the menu bar to arrange the selected nodes in a variety of alignments, namely, Left, Center, Right, Top, Middle, and Bottom.
 - Note** The use of **Distribute Nodes** and **Alignment** is mutually exclusive.
- When no nodes are selected, use the **Edit > Layout Nodes** from the menu bar 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 17 for information on using these tools.

What to Do Next

Create connections and interfaces.

Create Connections and Interfaces

Before You Begin

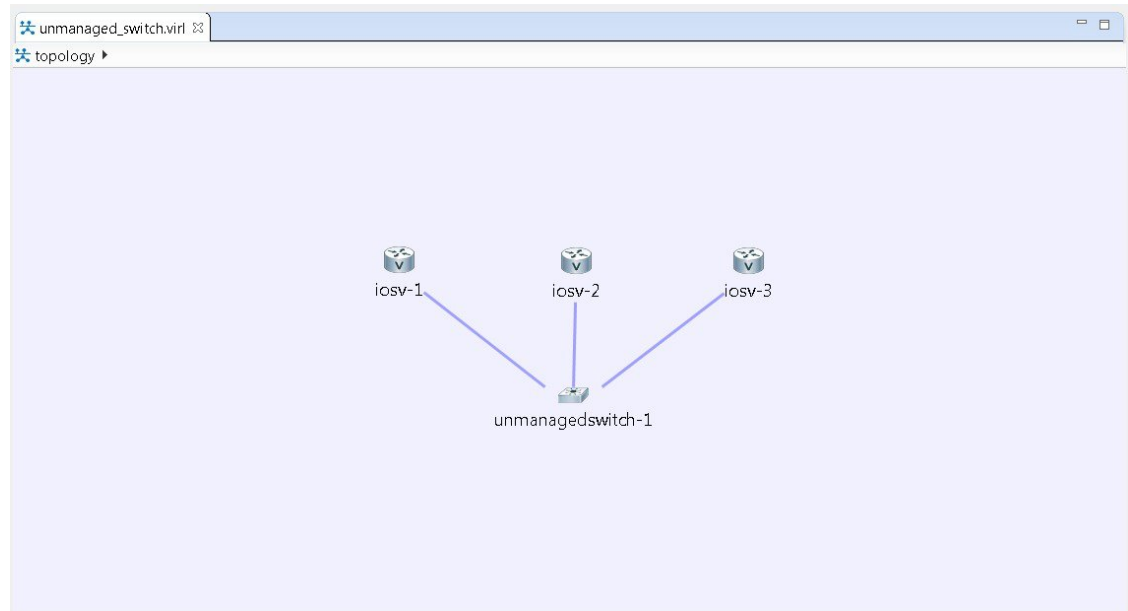
Nodes must be in place on the canvas of the **Topology Editor**.

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

Use Unmanaged Switches

Users have the choice to use either an unmanaged switch or Cisco IOSv layer 2 to provide a switching service. Unmanaged switches are used in place of the multipoint connection which was available in previous versions of Cisco Modeling Labs. For example, in the following figure there are 3 IOSv instances connected to an unmanaged switch. Each of the interfaces on iosv-1, iosv-2, and iosv-3 appear to be on the same subnet for point to point communication through the unmanaged switch.

Figure 50: Using an Unmanaged Switch in a Simple Topology



Unmanaged switch instances use the underlying Linux bridge process running under OpenStack control to provide this connectivity between the various virtual machines. It is a transparent switch.

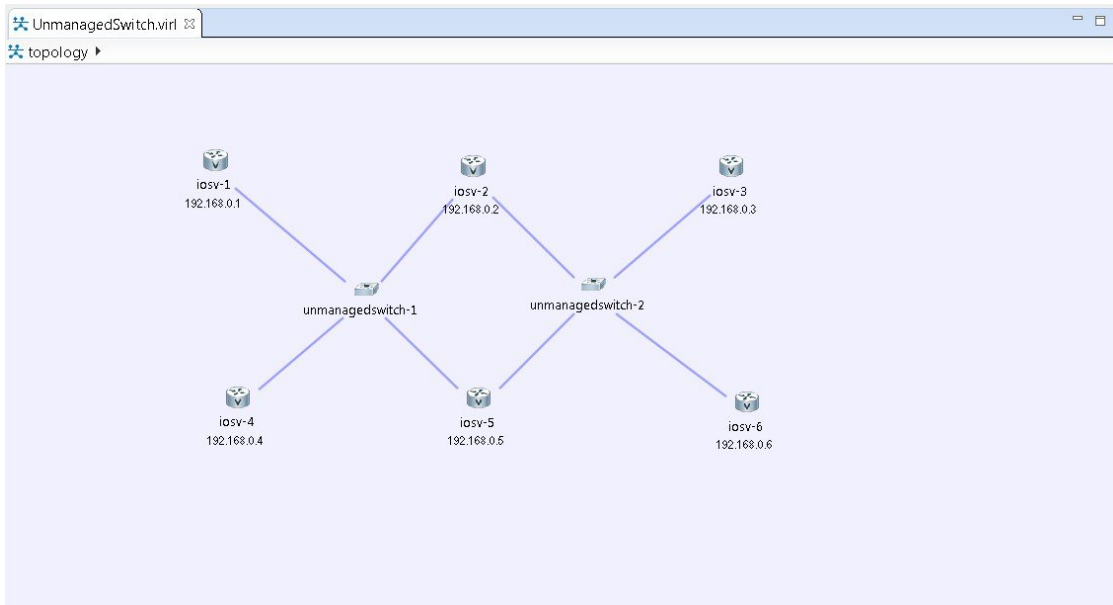
Before You Begin

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 **Nodes** view, click **Unmanaged Switch**.
 - Step 2** Click the area on the canvas where you want the unmanaged switch to appear.
 - Step 3** In the **Tool** view, click **Connect**.
 - Step 4** On the canvas, click the unmanaged switch node then click an end node. A connection appears.

Continue clicking unmanaged switch-node combinations until all connections are made.

Figure 51: Using an Unmanaged Switch



The Cisco IOSvL2 Switch Image

The Cisco IOSvL2 switch image is available in Cisco Modeling Labs, Release 1.1. The Cisco IOSvL2 switch image is a virtual machine like Cisco IOSv, Cisco IOS XRv, and so on. It runs and is configured in the same way as all other virtual machines. It runs a Cisco IOSv 15.2 switch image.



Note All Cisco IOSvL2 switch images in a topology are counted against the node limit.

A Cisco IOSvL2 switch image provides sixteen Gigabit Ethernet interfaces, reserving Gi0/0 for OOB management. It can be configured manually or using AutoNetkit.

Cisco IOSvL2 switch instances can operate in:

- Layer 2 mode
- Layer 3 mode

By default the instances operate in layer 3 mode. However, the primary use of the Cisco IOSvL2 switch image is for switching purposes.

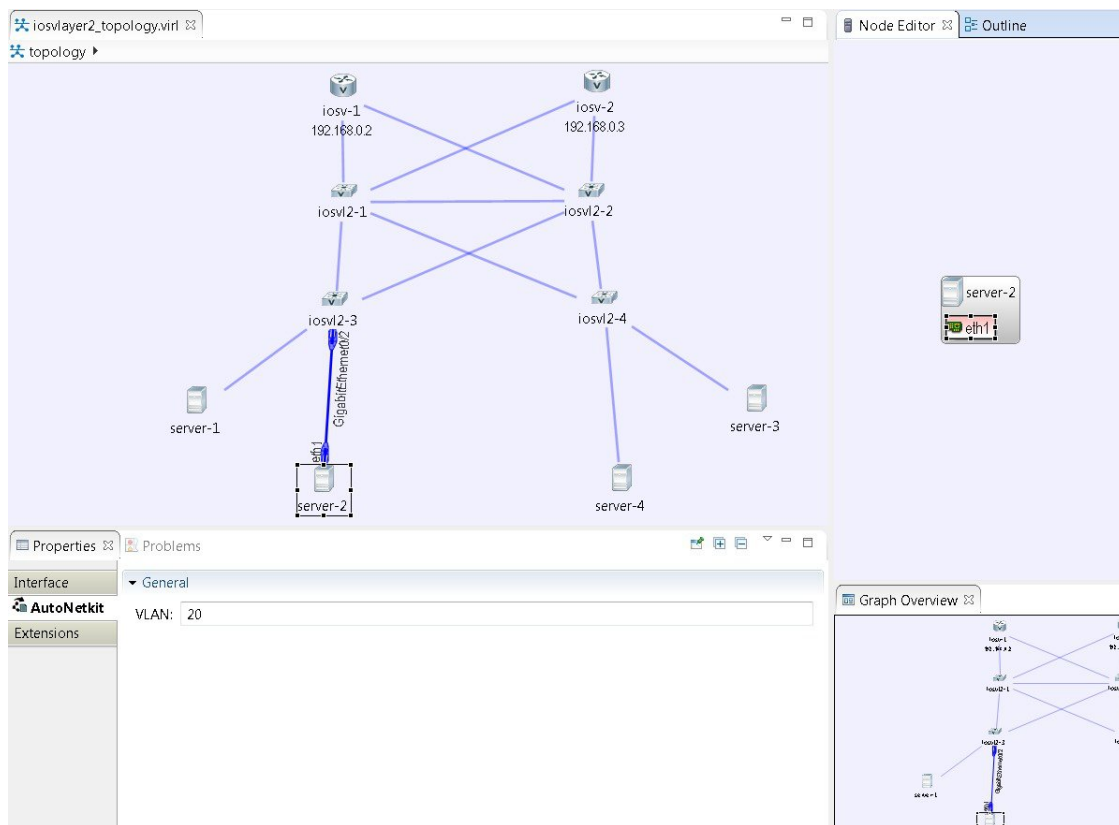
Any routers set up to connect to the Cisco IOSvL2 switch will be in switchport access mode. By default, all routers are placed in VLAN 2. You can specify which VLAN to place a port in by setting a VLAN attribute on the router interface. See [Assign VLANs](#), on page 104 for details on how to do this.

When using a switch to switch connection, it defaults to a 802.1q trunk port between the switches when configured using AutoNetkit.

Use the Cisco IOSvL2 Switch Image

- Step 1** In the **Nodes** view, click **IOSvL2**.
- Step 2** Click the canvas at each point where you want to place an IOSvL2 node. You can also drag the nodes on the canvas to position them.
- Step 3** Add additional node types as required.
- Step 4** Use the **Connect** tool to create connections between the nodes.
- Step 5** To specify which VLAN to place a port in, select the interface for the host or router in the **Node Editor**.
- Step 6** Under **Properties** > **AutoNetkit**, enter a value for the **VLAN** field, as shown.

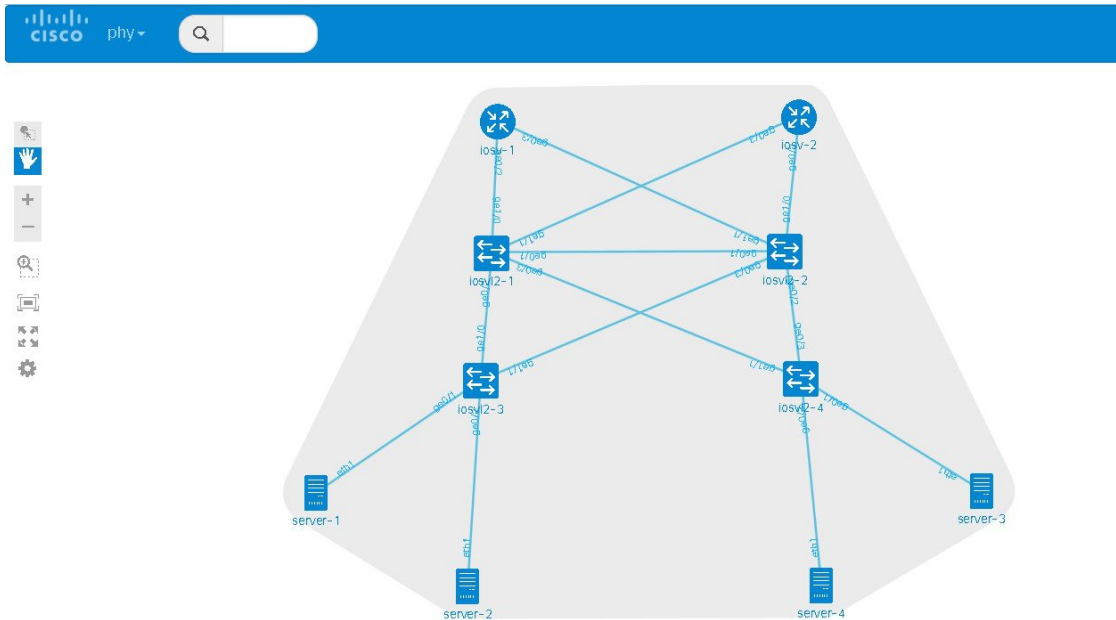
Figure 52: Setting the VLAN Attribute



- Step 7** From the toolbar, click **Build Initial Configurations** to generate a configuration for the topology using AutoNetkit. When prompted to open AutoNetkit visualization, click **Yes**.

AutoNetkit visualization for the topology opens in a browser window.

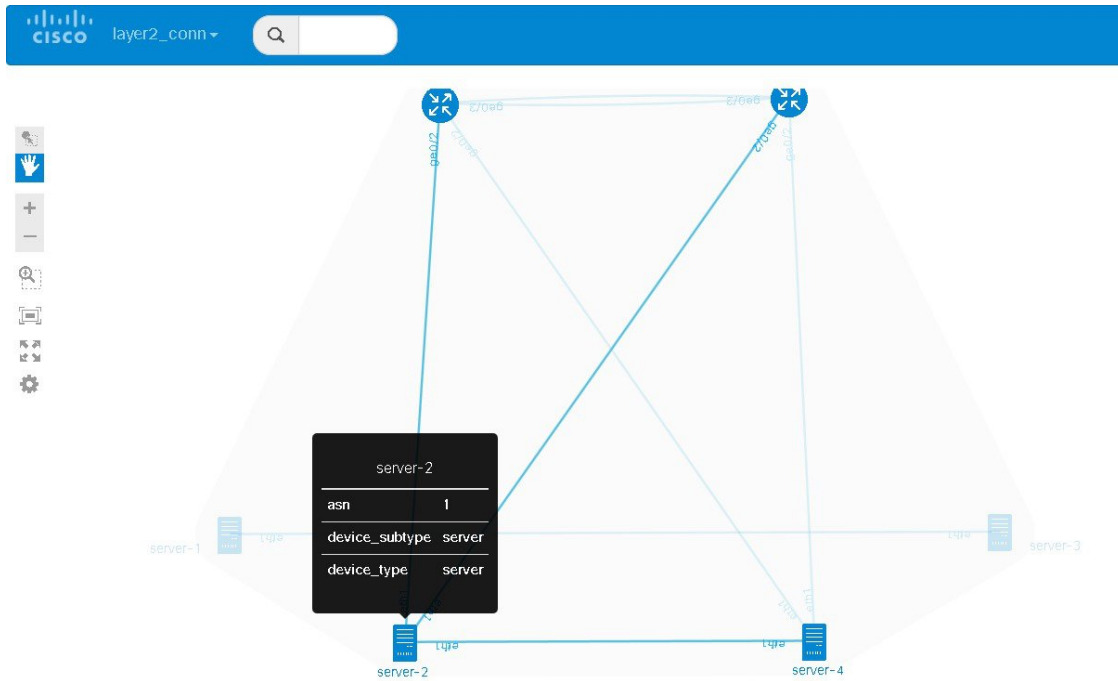
Figure 53: AutoNetkit Visualization



Step 8

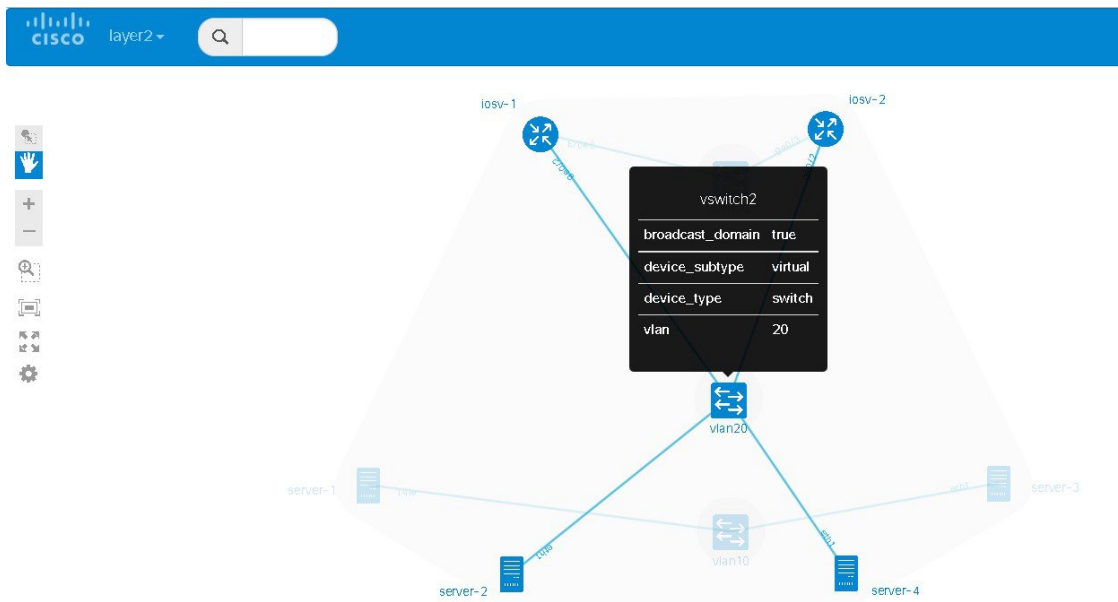
To view all broadcast domains that are enabled, select **layer2_conn** from the **phy** drop-down list. For example, hovering over server-2 in this example shows the routers and servers that are in the same VLAN. All other devices are greyed out.

Figure 54: Layer 2 Connectivity



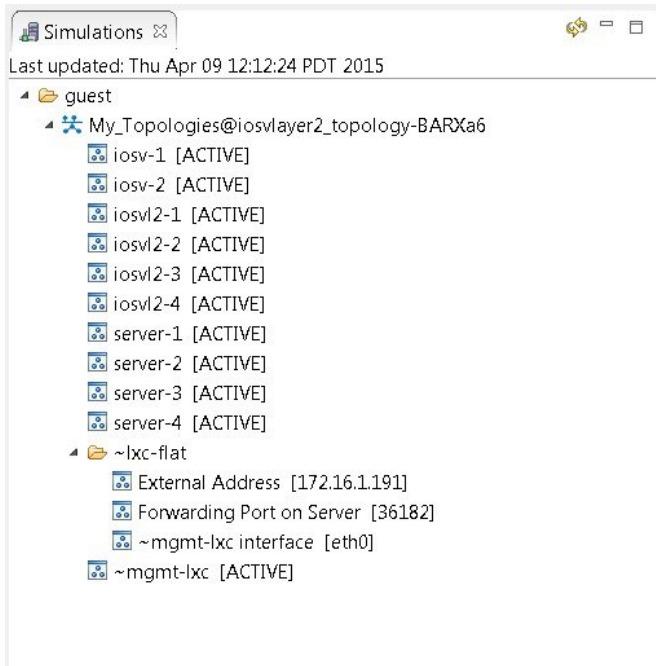
Selecting **layer2** from the **phy** drop-down list shows the switches representing the VLANs in place.

Figure 55: Layer 2 Switches



- Step 9** In the Cisco Modeling Labs client, click **Launch Simulation** to start the simulation. The simulation starts and is visible in the **Simulations** view.

Figure 56: Simulation with Cisco IOSvL2 Switch Image



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.

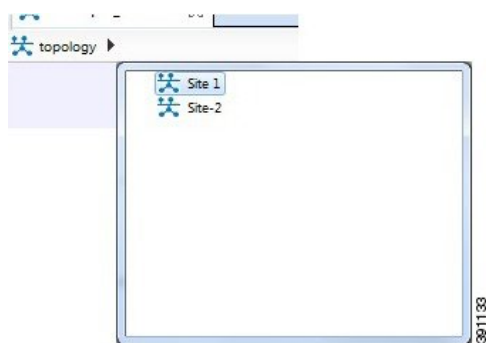
- From the **Edit** menu, choose **Group to Site**.
- Alternatively, you can drag the selected nodes to the site icon that you placed on the canvas.

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 57: 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:

- Select **Site 1**.
- Click **Connect**.
- Click the node that is the starting point of the connection, Site 1.
- Move the cursor to the breadcrumbs and click **Site 2**. Site 2 is displayed.
- 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:

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

Step 8 To delete a site from a topology:

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



Build a Configuration

- [Build a Configuration Overview, page 93](#)
- [Create and Modify a Node Configuration, page 93](#)
- [Create a Node Configuration Manually, page 94](#)
- [Use an Existing Node Configuration, page 95](#)
- [Import the Configuration from a Cariden MATE File, page 95](#)
- [Import the Configuration from a Visio vsdx File, page 99](#)
- [Create Node and Interface Configurations Using AutoNetkit, page 102](#)
- [Assign VLANs, page 104](#)
- [Use a Managed Switch, page 105](#)
- [Set Firewall Capabilities, page 110](#)
- [Set Security Levels, page 112](#)
- [Configure GRE Tunnels, page 114](#)

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.

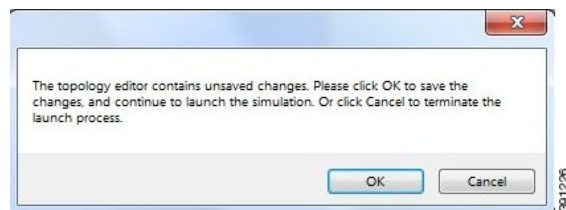
**Note**

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 the configuration based on these attributes** 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 58: Save Changes Before Launch



Create a Node Configuration Manually

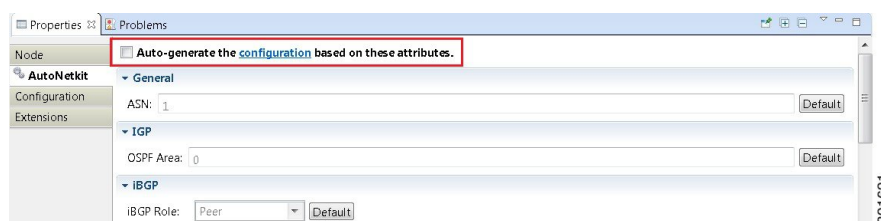
Before You Begin

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 59: Uncheck Auto-generate Check Box



Step 3 Click the **Configuration** tab.

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.

Note It is important to note that when generating configurations manually, you must create default logins with the username `cisco` and password `cisco`. These default login details are necessary when extracting your configurations at a later stage. See [Extract and Save Modified Configurations](#) for further information.

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 the **Configuration** tab.

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 or version 6.1.0. 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.

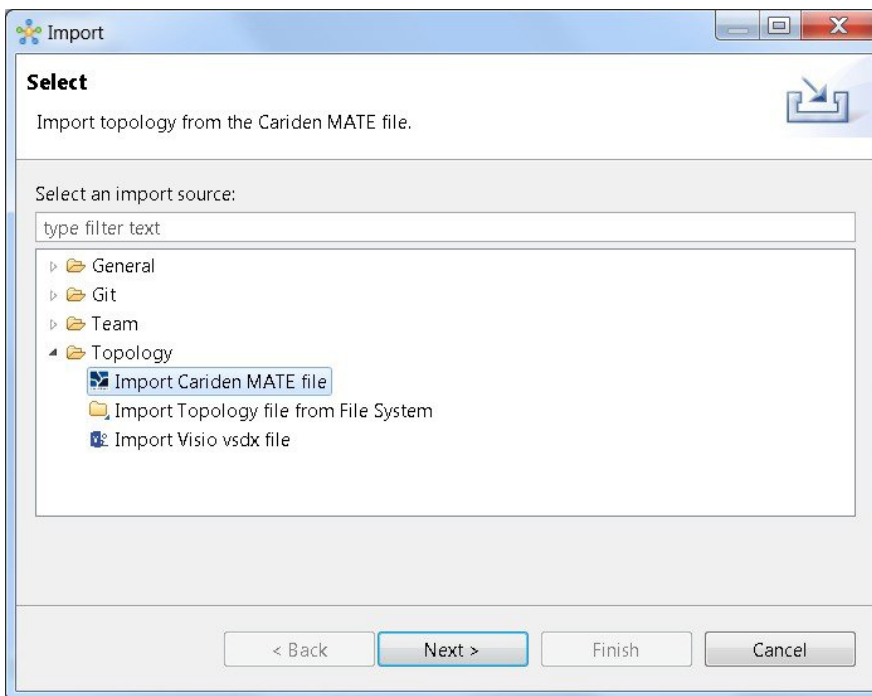
Before You Begin

- A valid Cariden MATE file is available on your file system.
- Cisco Modeling Labs client is running.

- Your license allows Cariden MATE file import.

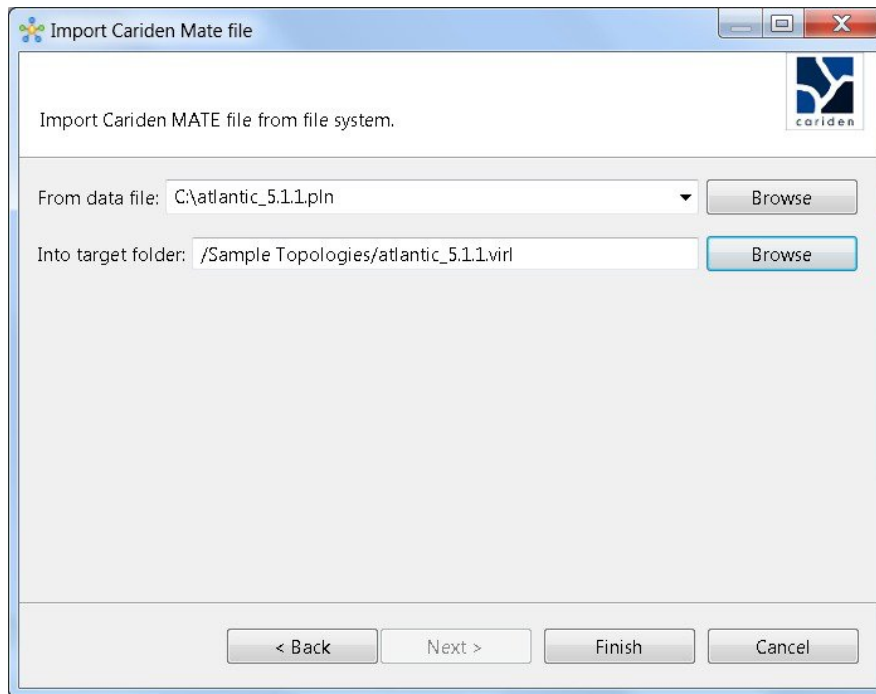
- Step 1** Choose **File > Import**.
A window appears, prompting you to Import Cariden MATE file.
- Step 2** Choose **Import Cariden MATE File** then click **Next**.

Figure 60: Import Cariden MATE File



- 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.

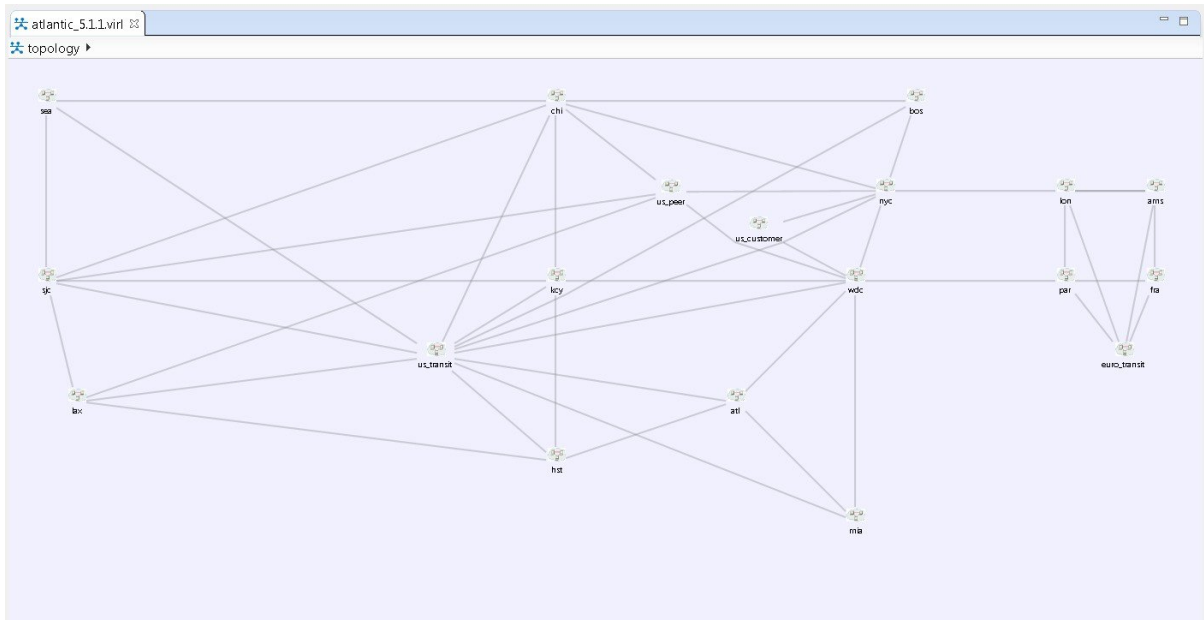
Figure 61: Choose the From and To Locations



- 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**.

The canvas opens and displays the topology.

Figure 62: Imported Cariden MATE File



Export the Configuration to Cariden MATE File

Before You Begin

- Cisco Modeling Labs client is running.
- A topology is open in the Topology Editor.
- Your license allows Cariden MATE file export.

-
- Step 1** Choose **File > Export**.
A window appears, prompting you to **Export to Cariden MATE file**.
- Step 2** Choose **Export Cariden MATE File** then click **Next**.
- Step 3** Choose the location **To file** for the Cariden MATE file export. Use **Browse** to select the target Project folder.
- Step 4** Enter a filename for the exported Cariden MATE file, or use the default filename. For example, **sample_topology.virl** is converted to **sample_topology.pln** and saved in the target directory.
- Step 5** Click **Finish**.
The Cisco Modeling Labs .virl file silently converts to a Cariden MATE .pln file.
-

Import the Configuration from a Visio vsdx File

You can import a topology from an existing Visio .vsdx file, version 2013 and later.

Before You Begin

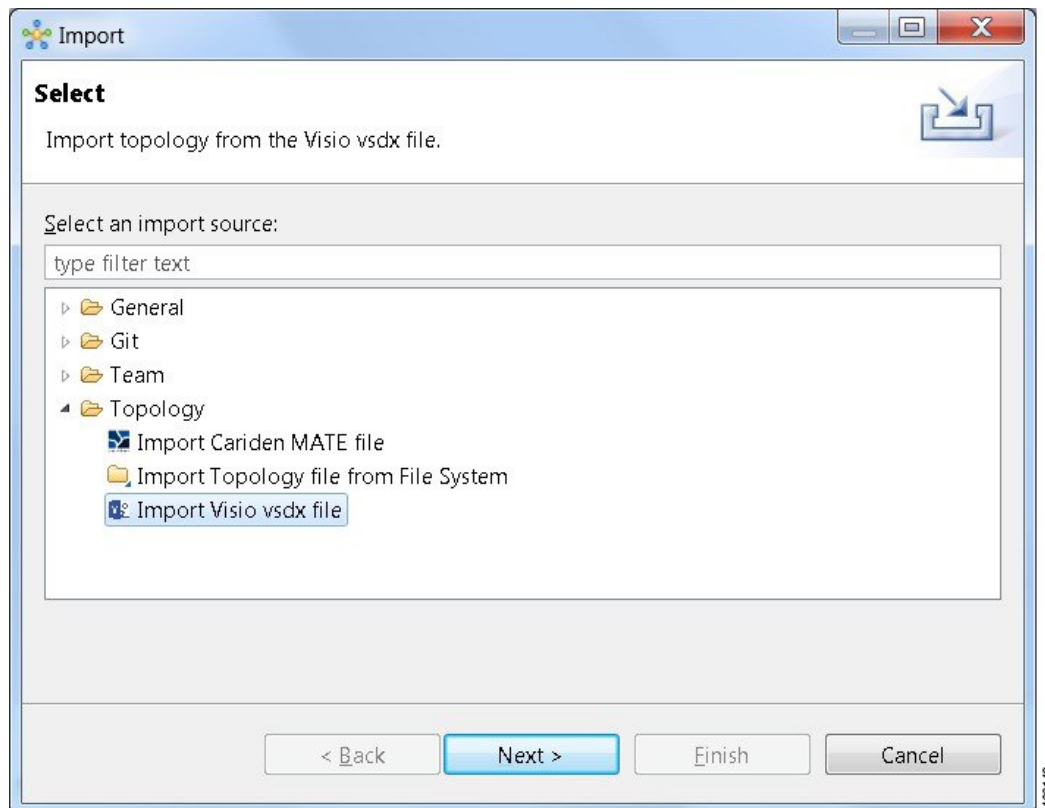
- A valid Visio file is available on your file system.
- Cisco Modeling Labs client is running.
- Your license allows Visio file import.

Step 1 Choose **File > Import**.

The **Import** dialog box appears.

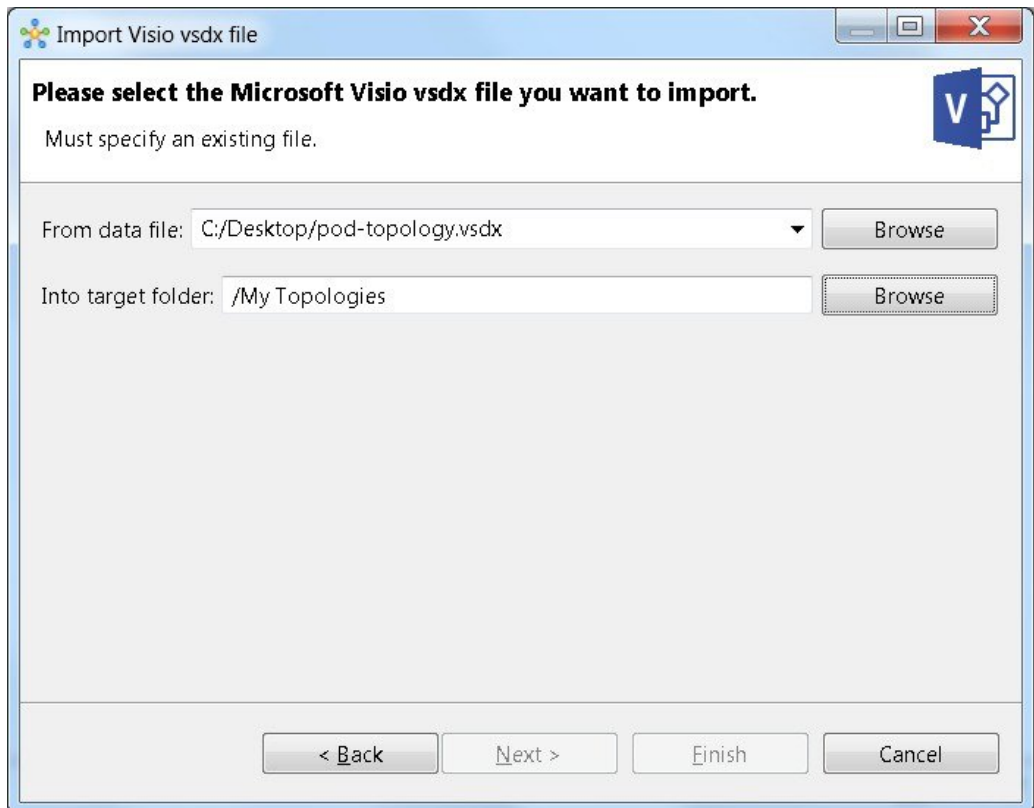
Step 2 Expand the **Topology** folder, choose **Import Visio vsdx File** and click **Next**.

Figure 63: Import Visio vsdx File



- Step 3** Choose the **From data file** Visio .vsdx file to import. Use **Browse** to select the directory and file to import.
- Step 4** Choose the location **Into target folder** for the Visio .vsdx file. Use **Browse** to select the target Project folder.

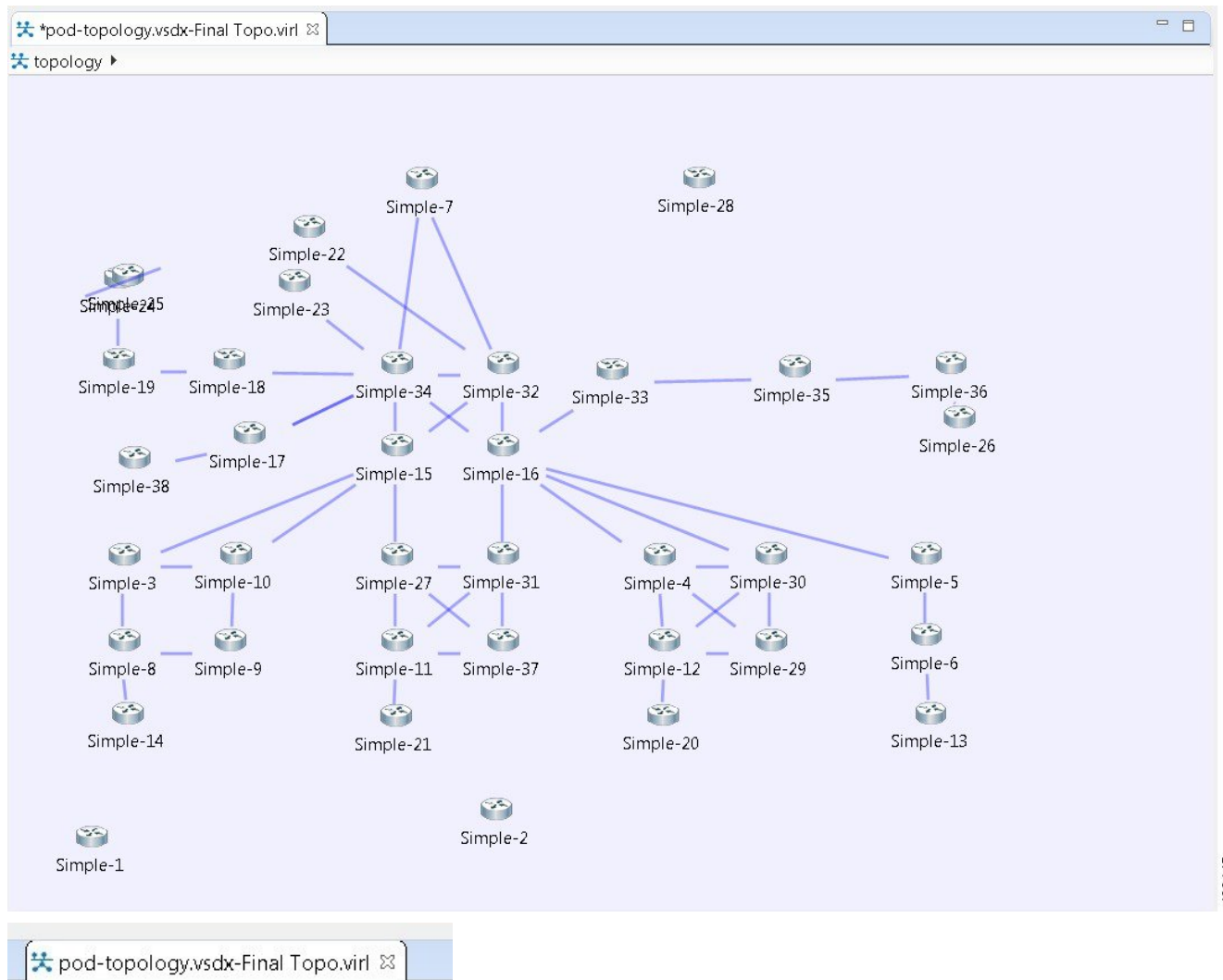
Figure 64: Choose the From and To Locations



- Step 5** Click **Finish**.

The Visio .vsdx file converts to a Cisco Modeling Labs .virl file, using the original filename of the file and it is automatically opened on the canvas.

Figure 65: Imported Visio .vsdx File



Note In this example, the file **pod-topology.vsdX** has been renamed by Cisco Modeling Labs to **pod-topology.vsdX-Final Topo.virl**. We recommend that for your .vsdx file imports, you rename the file(s) replacing the dot in .vsdx with '_'. In this example, **pod-topology.vsdX-Final Topo.virl** becomes **pod-topology_vsdX-Final Topo.virl**. You must do this as in Cisco Modeling Labs, the roster will parse the extra dot as a hierarchy delimiter and the simulation will fail.

Export the Configuration to SVG Files

For this release of Cisco Modeling Labs, export to Visio .vsdx files is not supported. However, export to .svg files is supported, as Visio supports the use of .svg files. The **Export** option can be used to export .virl files as .svg files.

Before You Begin

- Cisco Modeling Labs client is running.
- A topology is open in the Topology Editor.
- Your license allows SVG file export.

-
- Step 1** Choose **File > Export**.
A window appears, prompting you to **Export to SVG file**.
- Step 2** Choose **Export to SVG file** then click **Next**.
- Step 3** Choose the location **To file** for the SVG file export. Use **Browse** to select the target Project folder.
- Step 4** Enter a filename for the exported SVG file, or use the default filename. For example, **sample_topology.virl** is converted to **sample_topology.svg** and saved in the target directory.
- Step 5** Click **Finish**.
The Cisco Modeling Labs .virl file silently converts to a SVG .svg file.
-

Create Node and Interface Configurations Using AutoNetkit

Before You Begin

The topology design should be complete.

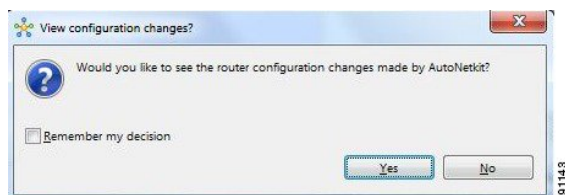
-
- Step 1** Verify the configuration for each node in the topology.
- a) In the **Topology Editor**, click a node.
 - b) In the **Properties** view, click **AutoNetkit**. Verify **Auto-generate the configuration based on these attributes** is checked or unchecked, depending on whether AutoNetkit will generate a configuration for that node.
Note Any pre-existing configuration for this node is overwritten when you choose **Build Initial Configurations** from the toolbar. Uncheck the **Auto-generate the configuration based on these attributes** 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 **Build Initial Configurations** from the toolbar. Alternatively, from the menu bar, choose **Configuration > Build Initial Configurations**. You are prompted to save any changes made since the previous configuration update.

If the **Auto-generate the configuration based on these attributes** 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.

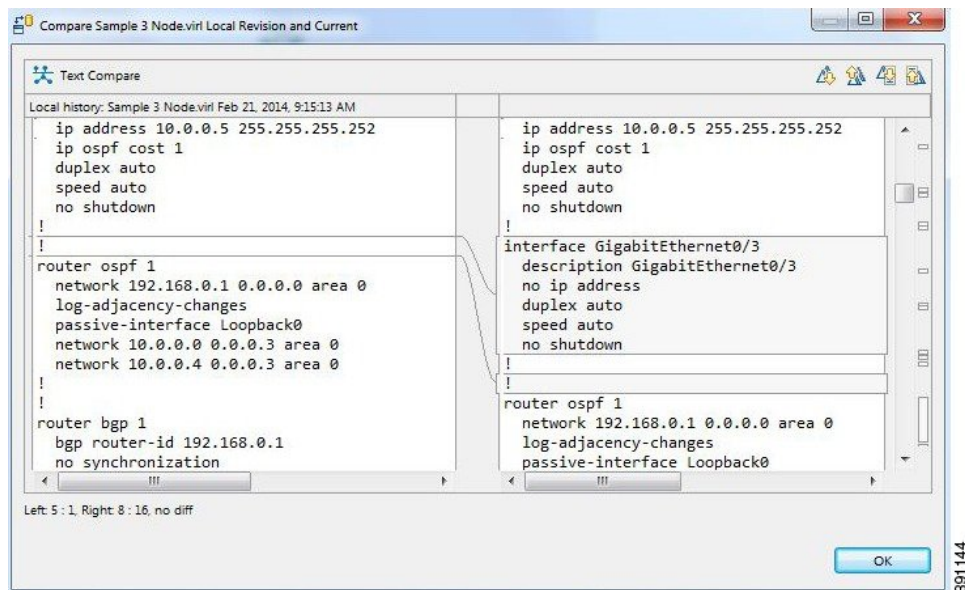
Figure 66: View Configuration Changes? Notification



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 > General > Reset All "Remember my decision" Dialog Boxes**.

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 67: Show Configuration Comparison Side-by-Side



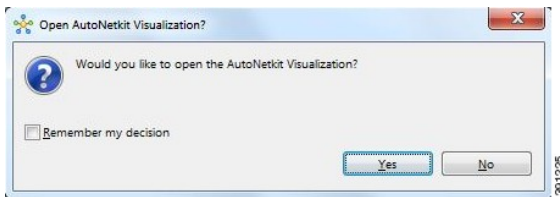
Click **OK** to close the comparison view.

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](#).

Figure 68: 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**.

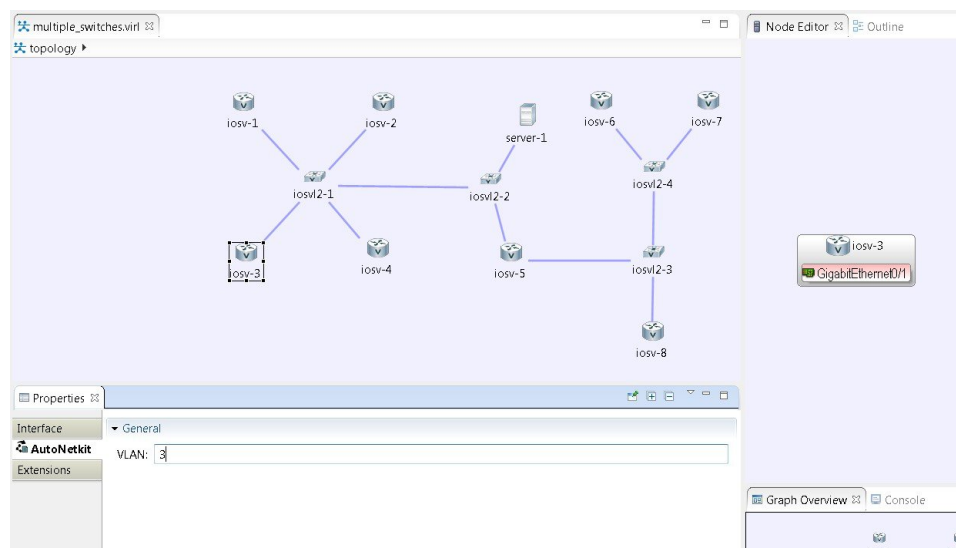
Assign VLANs

VLANs can be assigned to the interfaces of the end nodes, using the **Properties > Interface** view.

VLANs are set using the **VLAN** property under the **General** tab in the **AutoNetkit** field on the interface. The interface is selected in the **Node Editor**. The properties are set on the interfaces of the nodes connected to the IOSvL2 image, such as on the IOSv nodes, server node interfaces.

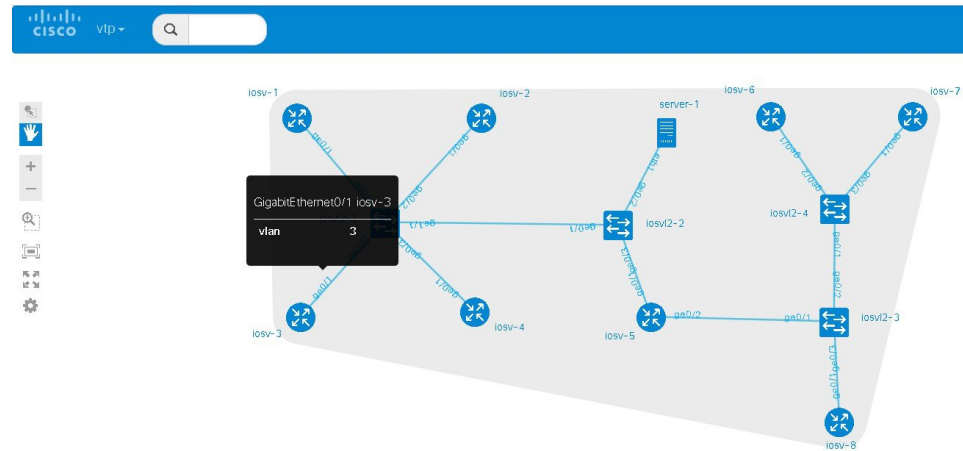
The following example shows how to set a VLAN property.

Figure 69: Set a VLAN Property



These VLAN values are displayed in the VLAN attribute of the interfaces in AutoNetkit visualization:

Figure 70: VLAN Property Assigned



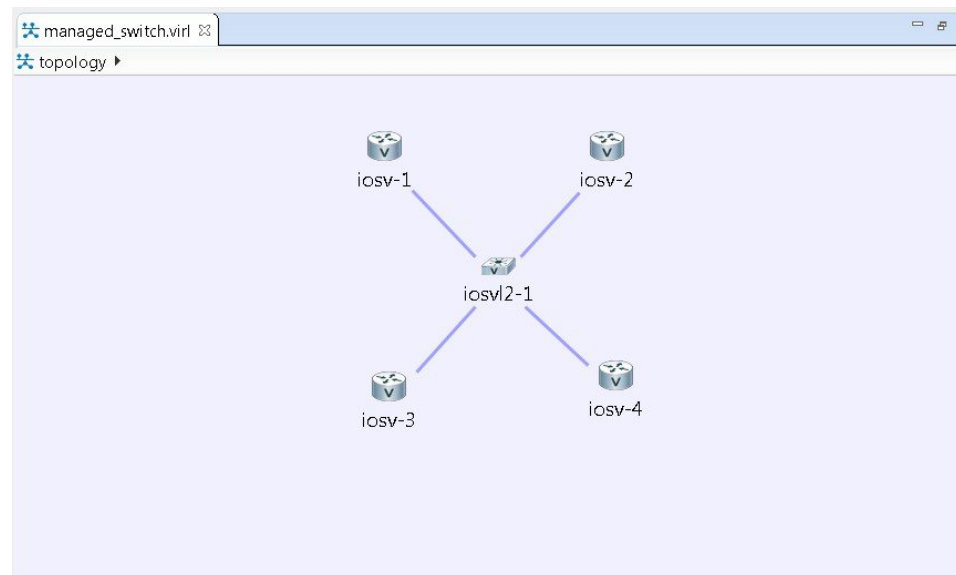
Use a Managed Switch

The Cisco IOSv Layer 2 switch introduces a managed switch to the Cisco Modeling Labs environment.

By default, all VLANs are placed in `vlan2`.

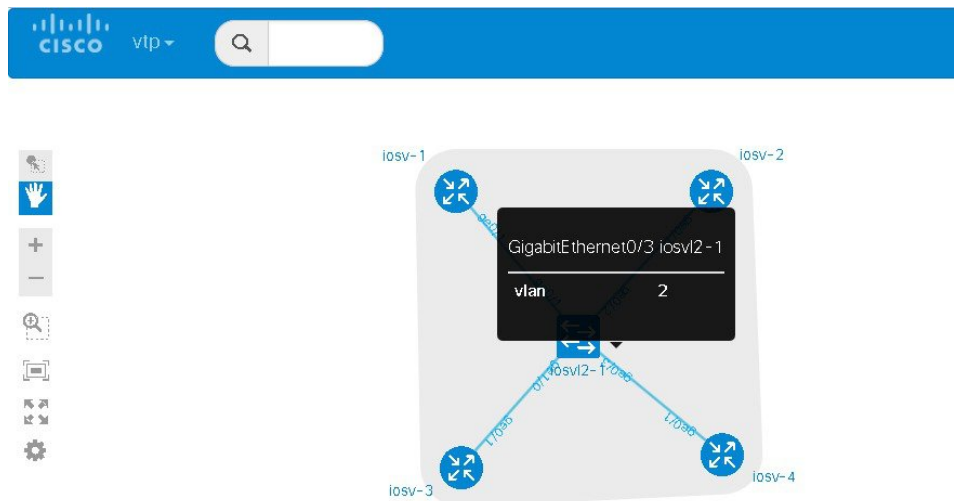
For example, consider the following topology which includes four nodes and one IOSvL2 image:

Figure 71: Using a Managed Switch



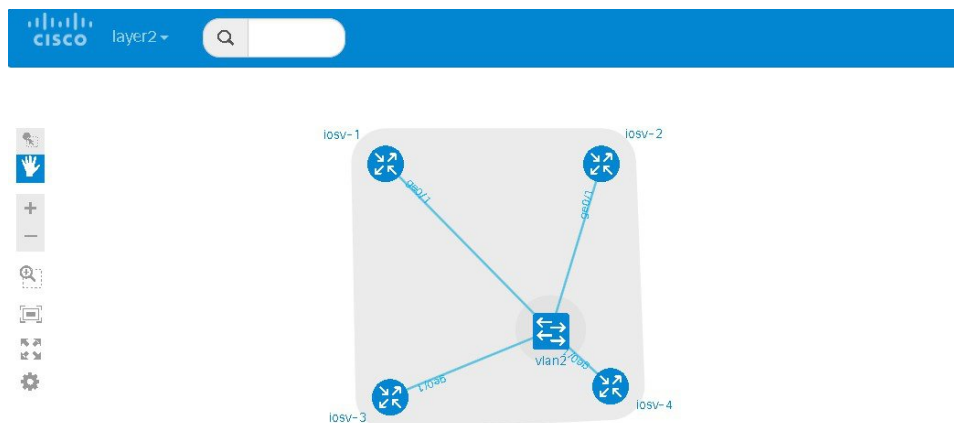
After running AutoNetkit, you can see the default VLAN assigned using the **vtp** view:

Figure 72: VLAN Assignment



The **layer2** view shows the vtp domain originating from the virtual switch for vlan2:

Figure 73: Vtp Domain - layer2.tiff



The relevant configuration for the IOSvL2 image is:

```
interface GigabitEthernet0/1
description to iosv-1
switchport access vlan 2
switchport mode access
no shutdown
!
interface GigabitEthernet0/2
description to iosv-3
switchport access vlan 2
switchport mode access
```

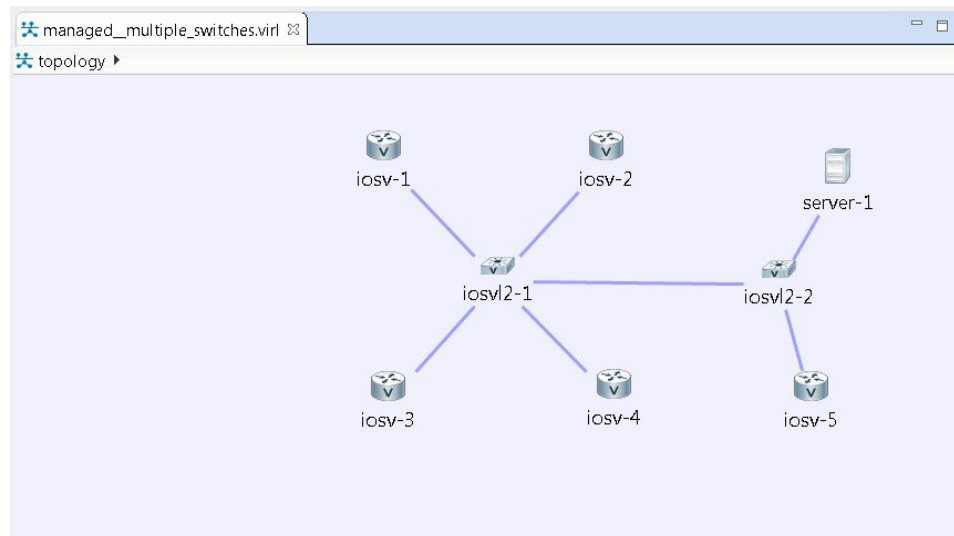
```
no shutdown
!  
interface GigabitEthernet0/3  
description to iosv-2  
switchport access vlan 2  
switchport mode access  
no shutdown  
!  
interface GigabitEthernet1/0  
description to iosv-4  
switchport access vlan 2  
switchport mode access  
no shutdown  
!
```

Use Multiple Managed Switches

It is permissible to connect multiple managed switches together. Multiple managed switches connected together form a trunk link between the switches and their appropriate vtp domains.

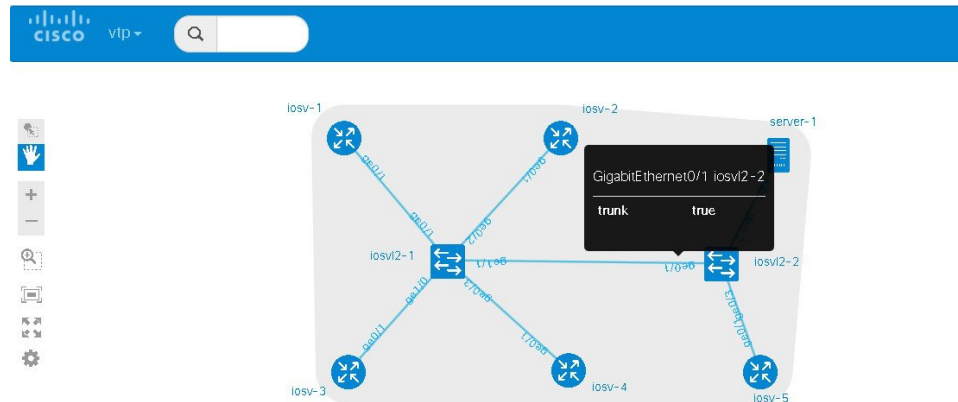
In the following example, two managed switches are connected together:

Figure 74: Using Multiple Managed Switches



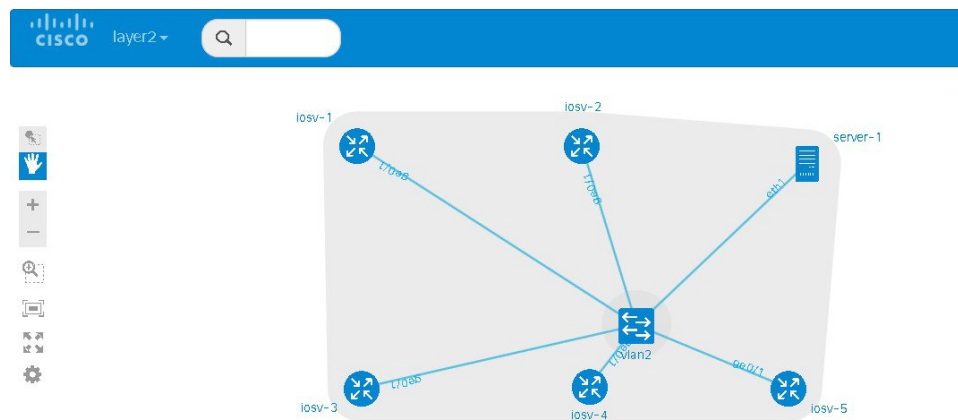
The **vtp** view shows the trunk link created between the two managed switches:

Figure 75: Trunk Link Created



The **layer2** view shows the resulting layer2 connectivity, where both of the managed switches have been aggregated into a single vtp domain for the default vlan2:

Figure 76: Layer2 Connectivity



The relevant configurations for iosv2-1 and iosv2-2 on the trunk port are shown below.

iosv2-1

```
interface GigabitEthernet0/1
description to iosv-1
switchport access vlan 2
switchport mode access
no shutdown
!
interface GigabitEthernet0/2
description to iosv-3
switchport access vlan 2
switchport mode access
no shutdown
```

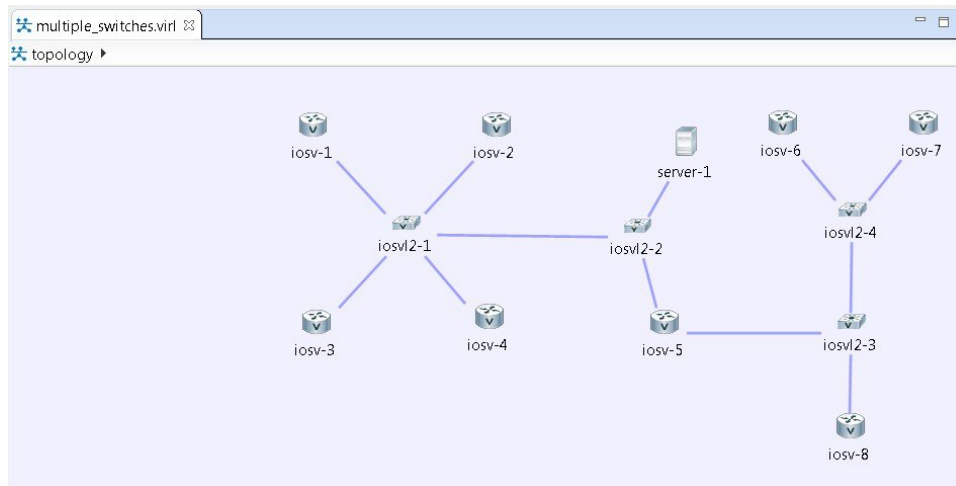
```
!  
interface GigabitEthernet0/3  
  description to iosv-2  
  switchport access vlan 2  
  switchport mode access  
  no shutdown  
!  
interface GigabitEthernet1/0  
  description to iosv-4  
  switchport access vlan 2  
  switchport mode access  
  no shutdown  
!  
interface GigabitEthernet1/1  
  description to iosvl2-2  
  switchport trunk encapsulation dot1q  
  switchport mode trunk  
  no shutdown  
!  
iosvl2-2  
interface GigabitEthernet0/1  
  description to iosvl2-1  
  switchport trunk encapsulation dot1q  
  switchport mode trunk  
  no shutdown  
!  
interface GigabitEthernet0/2  
  description to server-1  
  switchport access vlan 2  
  switchport mode access  
  no shutdown  
!  
interface GigabitEthernet0/3  
  description to iosv-5  
  switchport access vlan 2  
  switchport mode access  
  no shutdown  
!
```

Use Multiple Unconnected Managed Switches

In cases where there are multiple managed switches, only those that are directly connected, either through a point-to-point link or via an unmanaged switch are connected.

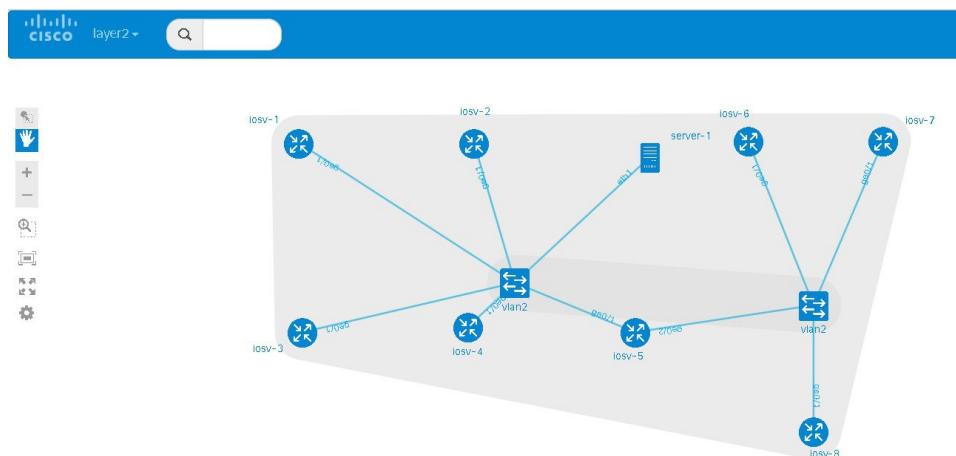
For example, in this topology, there are two sets of managed switches:

Figure 77: Using Multiple Managed Switches



After running AutoNetkit, the resulting **layer2** view shows two separate layer 2 domains:

Figure 78: Separate Layer2 Domains



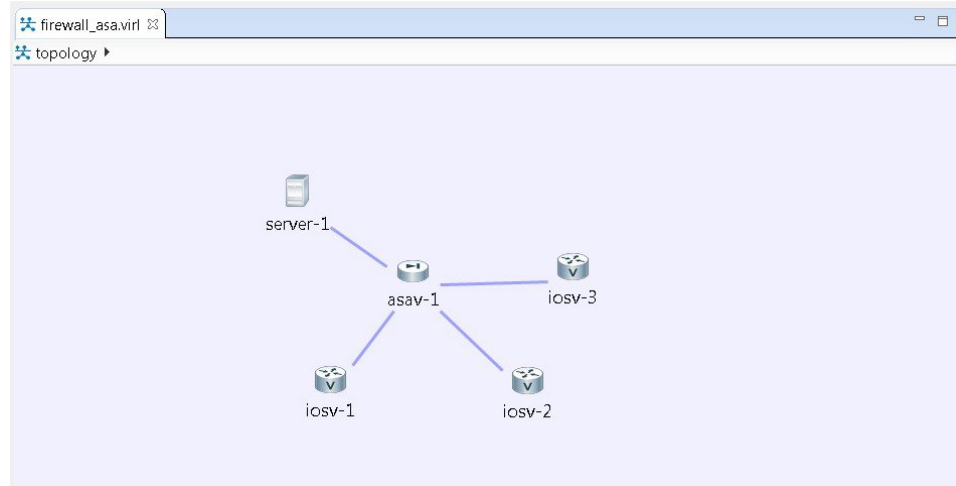
Set Firewall Capabilities

For this release of Cisco Modeling Labs, the Cisco ASAv image is available to purchase separately. The Cisco ASAv image adds firewall capabilities to Cisco Modeling Labs.

The default AutoNetkit configuration puts each interface into security-level 0, adds a nameif, and allows http, SSH, and Telnet access to this nameif.

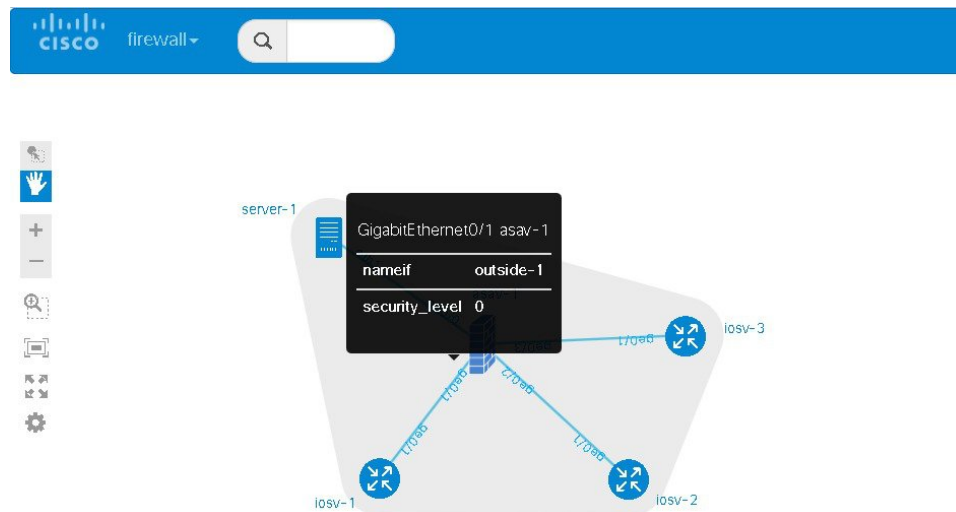
For example, consider the following topology which includes three IOSv nodes, one server node and one ASAv node:

Figure 79: Example Topology Showing a Cisco ASAv Node



After running AutoNetkit, the firewall view shows the allocated properties on the interfaces:

Figure 80: Allocated Firewall Properties



The configuration for the interface is:

```

interface GigabitEthernet0/0
description to server-1
nameif outside
security-level 0
no shutdown
ip address 10.0.0.5 255.255.255.252
interface GigabitEthernet0/1
description to iosv-1
  
```

```

nameif outside-1
security-level 0
no shutdown
ip address 10.0.0.9 255.255.255.252
interface GigabitEthernet0/2
description to iosv-2
nameif outside-2
security-level 0
no shutdown
ip address 10.0.0.13 255.255.255.252
interface GigabitEthernet0/3
description to iosv-3
nameif outside-3
security-level 0
no shutdown
ip address 10.0.0.17 255.255.255.252

```

The access details are:

```

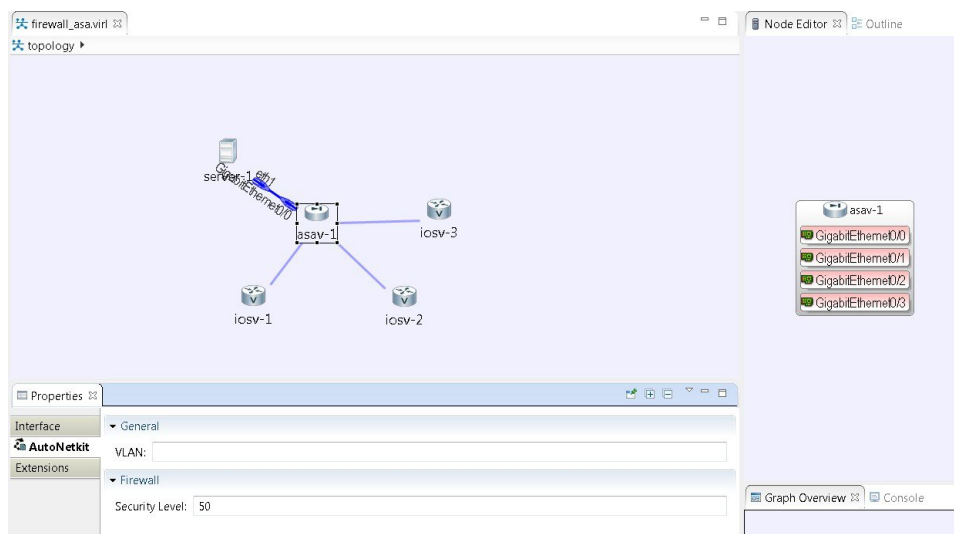
http 0.0.0.0 0.0.0.0 mgmt
ssh 0.0.0.0 0.0.0.0 mgmt
telnet 0.0.0.0 0.0.0.0 mgmt
http 0.0.0.0 0.0.0.0 outside
ssh 0.0.0.0 0.0.0.0 outside
telnet 0.0.0.0 0.0.0.0 outside
http 0.0.0.0 0.0.0.0 outside-1
ssh 0.0.0.0 0.0.0.0 outside-1
telnet 0.0.0.0 0.0.0.0 outside-1
http 0.0.0.0 0.0.0.0 outside-2
ssh 0.0.0.0 0.0.0.0 outside-2
telnet 0.0.0.0 0.0.0.0 outside-2
http 0.0.0.0 0.0.0.0 outside-3
ssh 0.0.0.0 0.0.0.0 outside-3
telnet 0.0.0.0 0.0.0.0 outside-3

```

Set Security Levels

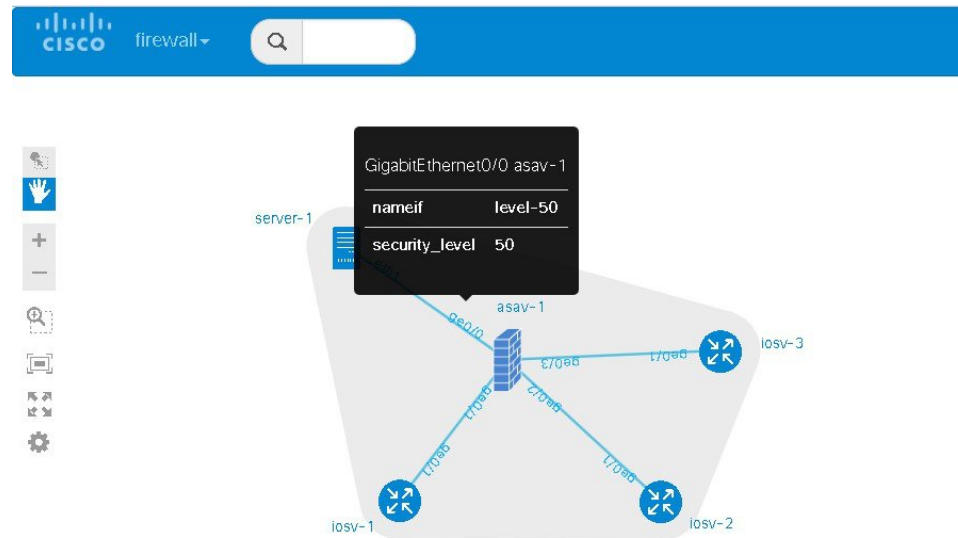
Security levels are set using the **Security Level** property under the **Firewall** tab in the **AutoNetkit** field on the interface. The interface is selected in the **Node Editor**. The properties are set on the Cisco ASA node's interfaces, as shown.

Figure 81: Set the Security Level



The security level is displayed in the `security_level` attribute of the interfaces in AutoNetkit visualization:

Figure 82: Security Level Attribute Set



The configuration for `nameif` is updated.

```
interface GigabitEthernet0/0
  description to server-1
  nameif level-50
  security-level 50
  no shutdown
  ip address 10.0.0.5 255.255.255.252
interface GigabitEthernet0/1
  description to iosv-1
  nameif outside
  security-level 0
  no shutdown
  ip address 10.0.0.9 255.255.255.252
interface GigabitEthernet0/2
  description to iosv-2
  nameif outside-1
  security-level 0
  no shutdown
  ip address 10.0.0.13 255.255.255.252
interface GigabitEthernet0/3
  description to iosv-3
  nameif outside-2
  security-level 0
  no shutdown
  ip address 10.0.0.17 255.255.255.252
```

The access details are also updated.

```
http 0.0.0.0 0.0.0.0 level-50
ssh 0.0.0.0 0.0.0.0 level-50
telnet 0.0.0.0 0.0.0.0 level-50
http 0.0.0.0 0.0.0.0 mgmt
ssh 0.0.0.0 0.0.0.0 mgmt
telnet 0.0.0.0 0.0.0.0 mgmt
http 0.0.0.0 0.0.0.0 outside
ssh 0.0.0.0 0.0.0.0 outside
telnet 0.0.0.0 0.0.0.0 outside
http 0.0.0.0 0.0.0.0 outside-1
ssh 0.0.0.0 0.0.0.0 outside-1
```

```
telnet 0.0.0.0 0.0.0.0 outside-1
http 0.0.0.0 0.0.0.0 outside-2
ssh 0.0.0.0 0.0.0.0 outside-2
telnet 0.0.0.0 0.0.0.0 outside-2
```



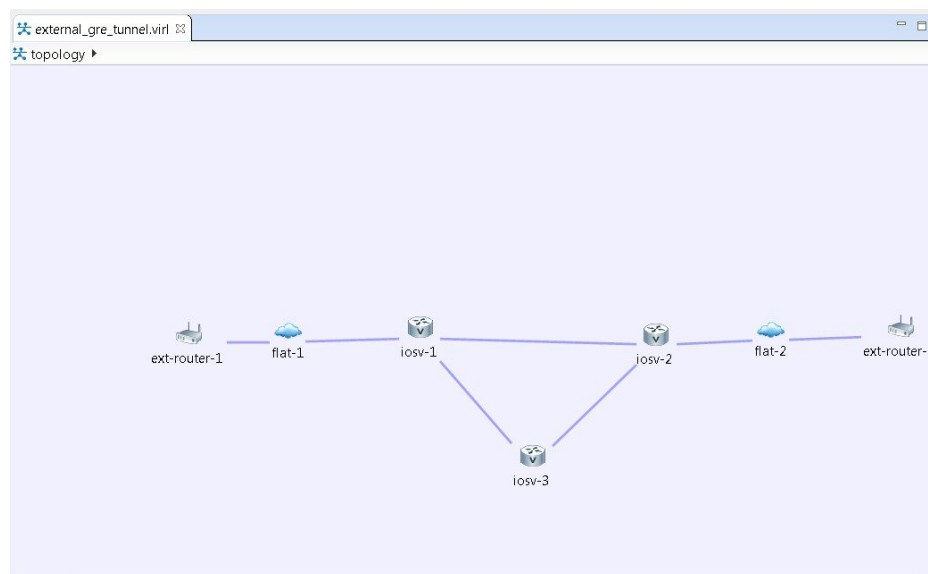
Note AutoNetkit automatically renames the nameif if there are multiple interfaces with the same security level.

Configure GRE Tunnels

Generic routing encapsulation (GRE) is a simple IP packet encapsulation protocol that is used to transport packets over a network. Information is sent from one network to the other through a GRE tunnel.

The GRE tunnel functionality uses the IOSv subtype as the GRE tunnel head and connects from an IOSv instance out over the FLAT/FLAT1/SNAT connector to some other device which is the far-end of the GRE Tunnel.

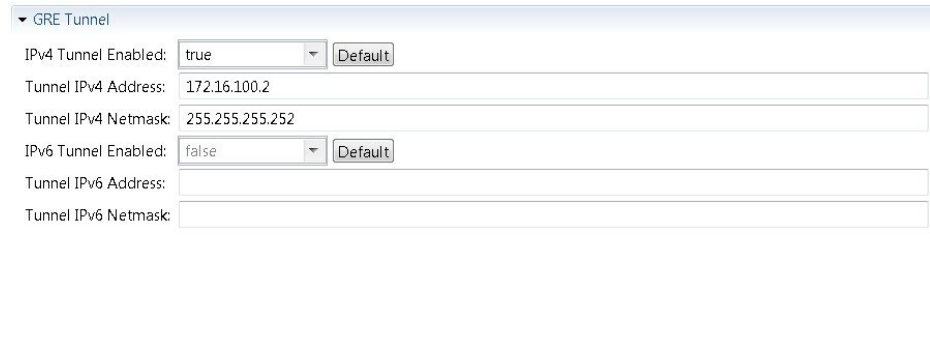
Figure 83: Using GRE Tunnels



In this example, you set the values on node iosv-1 and node iosv-2 to tell AutoNetkit to create the configuration for a GRE tunnel terminating on the external router node, `ext_router_1`.

So on iosv-1, set the tunnel IP address and subnet mask of the far-end device ext-router-1. Similarly, on the ext-router-1, set the tunnel IP address and subnet mask of the far-end device iosv-1.

Figure 84: Tunnel IP Address and Subnet Mask for ext_router_1

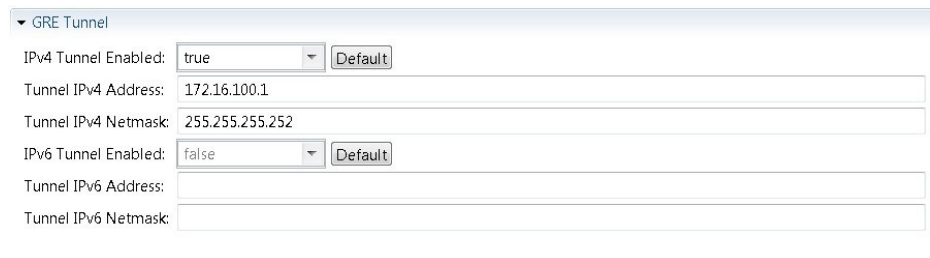


The screenshot shows the configuration for a GRE Tunnel on the device ext_router_1. The configuration is as follows:

Field	Value
GRE Tunnel	
IPv4 Tunnel Enabled:	true
Tunnel IPv4 Address:	172.16.100.2
Tunnel IPv4 Netmask:	255.255.255.252
IPv6 Tunnel Enabled:	false
Tunnel IPv6 Address:	
Tunnel IPv6 Netmask:	

405456

Figure 85: Tunnel IP Address and Subnet Mask for iosv-1



The screenshot shows the configuration for a GRE Tunnel on the device iosv-1. The configuration is as follows:

Field	Value
GRE Tunnel	
IPv4 Tunnel Enabled:	true
Tunnel IPv4 Address:	172.16.100.1
Tunnel IPv4 Netmask:	255.255.255.252
IPv6 Tunnel Enabled:	false
Tunnel IPv6 Address:	
Tunnel IPv6 Netmask:	

405458

On iosv-2, set the tunnel IP address and subnet mask of the far-end device ext-router-2. Similarly, on the ext-router-2, set the tunnel IP address and subnet mask of the far-end device iosv-2.

Figure 86: Tunnel IP Address and Subnet Mask for ext_router_2

GRE Tunnel configuration for ext_router_2:

- IPv4 Tunnel Enabled: true (Default)
- Tunnel IPv4 Address: 172.16.200.1
- Tunnel IPv4 Netmask: 255.255.255.252
- IPv6 Tunnel Enabled: false (Default)
- Tunnel IPv6 Address:
- Tunnel IPv6 Netmask:

405467

Figure 87: Tunnel IP Address and Subnet Mask for iosv-2

GRE Tunnel configuration for iosv-2:

- IPv4 Tunnel Enabled: true (Default)
- Tunnel IPv4 Address: 172.16.200.2
- Tunnel IPv4 Netmask: 255.255.255.252
- IPv6 Tunnel Enabled: false (Default)
- Tunnel IPv6 Address:
- Tunnel IPv6 Netmask:

405469

When the configurations are built, AutoNetkit selects the appropriate corresponding IP address and applies it to the interface as follows:

```
!
interface Tunnell
 ip address 172.16.100.2 255.255.255.252
 tunnel source GigabitEthernet0/3
 tunnel destination 0.0.0.0
!
```

The tunnel destination is blank since it needs to be set to the IP address of the far-end device, which you may or may not know in advance. However, you can edit the configuration in the Cisco Modeling Labs client GUI before you start up the simulation. So if you do know the target address, you can add the target IP address in there (tunnel destination x.x.x.x.) Remember that it is not the IP address of the tunnel that goes in here but the IP address of the router/device terminating the GRE tunnel itself. If this is a devices that is on the FLAT network directly, then a 172.16.1.x address would go in here.

To make things simple and repeatable, you can use a static IP address on the interface of the IOSv GRE tunnel device that connects to the FLAT/FLAT1/SNAT connector.

Figure 88: Static IP Address for flat-1

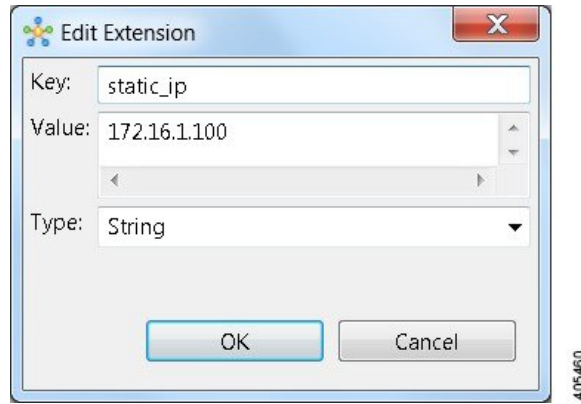
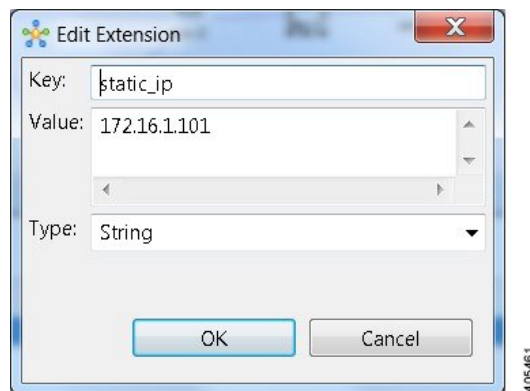


Figure 89: Static IP Address for flat-2



This provides a target address that the other device can then try to connect to and it is the same IP address each time the simulation is started.



Note

You cannot do this using the standard guest account. The simulation will fail as you are using a system-level resource (the Static IP address), so an account with administrative permissions is required.

You must create this account in the **User Workspace Management** interface.

In the **User Workspace Management** interface, under the **Projects** tab, click **Add** to create a new project, as follows:

Figure 90: Create a New Project

Create Project

General Settings

Name

Description

Expires

Enabled

Project Quotas

Instances

RAM (MB)

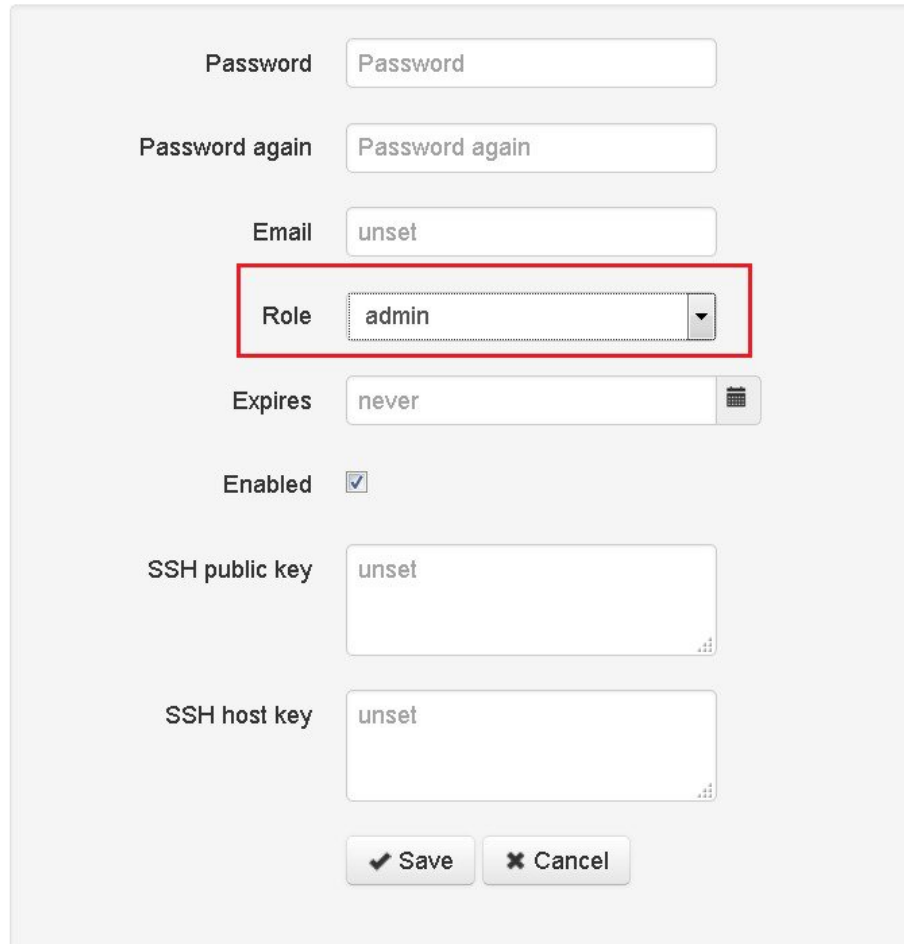
VCPUS

405462

In the corresponding user created for the project, set **Role** to **admin**.

Figure 91: Update the Role Field

Edit User *gre_tunneling*



The screenshot shows the configuration page for editing the user *gre_tunneling*. The fields are as follows:

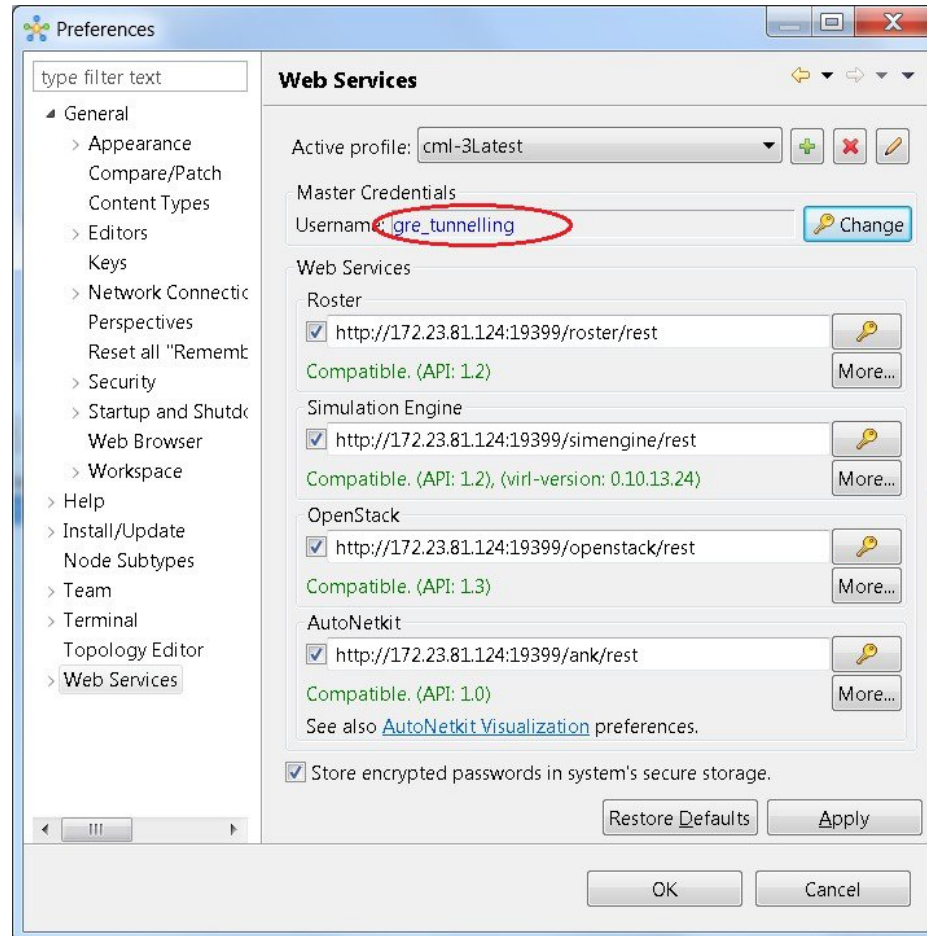
- Password:** Password
- Password again:** Password again
- Email:** unset
- Role:** admin (highlighted with a red box)
- Expires:** never
- Enabled:**
- SSH public key:** unset
- SSH host key:** unset

At the bottom, there are two buttons: **Save** (with a checkmark icon) and **Cancel** (with an 'X' icon).

405463

In the Cisco Modeling Labs client GUI, choose **File > Preferences > Web Services**. In the **Web Services** dialog box, click **Change** under Master Credentials to login with the newly created user.

Figure 92: Log In as New Role



You can now start your simulation.



Visualizing the Topology

- [AutoNetkit Visualization, page 121](#)

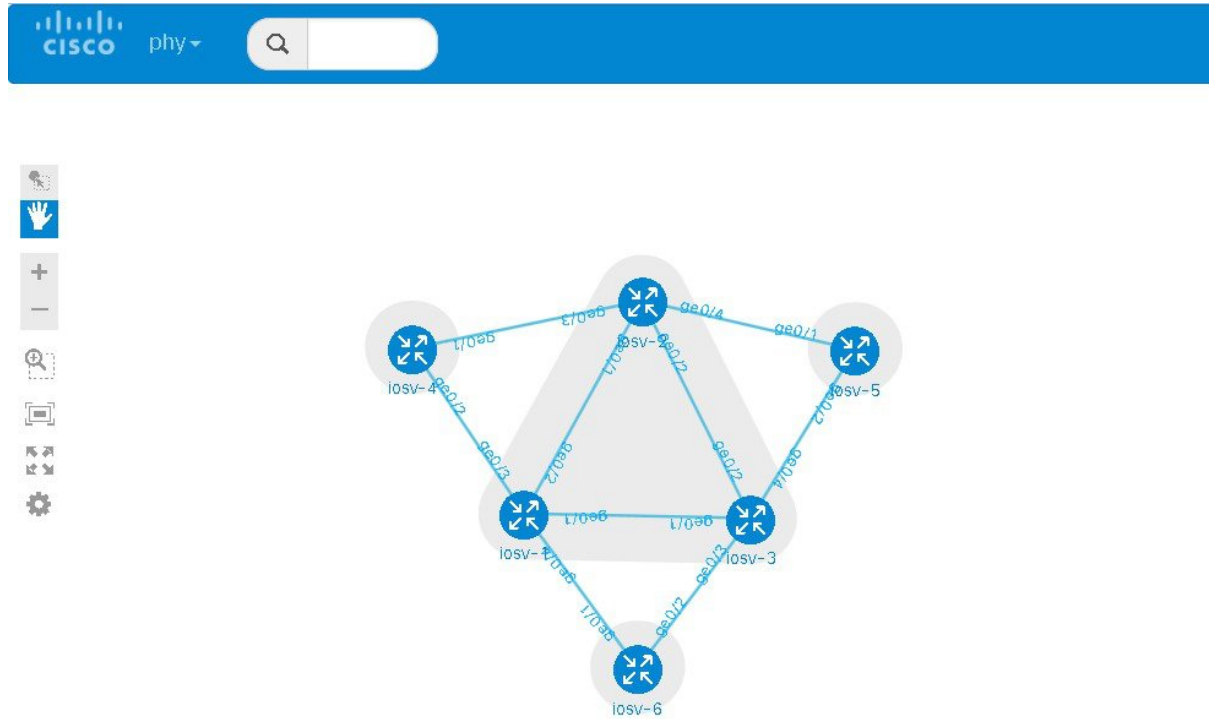
AutoNetkit Visualization

The AutoNetkit 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 AutoNetkit visualization phase, you must have designed the topology and generated the node configurations using parameters defined in AutoNetkit.

AutoNetkit visualization runs in a browser window, either within the Cisco Modeling Labs client or in a separate window browser. Ensure that you use a compatible browser, as described in the [Cisco Modeling Labs Corporate Edition System Administrator Installation Guide, Release 1.1](#) and [Cisco Modeling Labs Corporate Edition Client Installation Guide, Release 1.1](#), for the version of Cisco Modeling Labs that you are using.

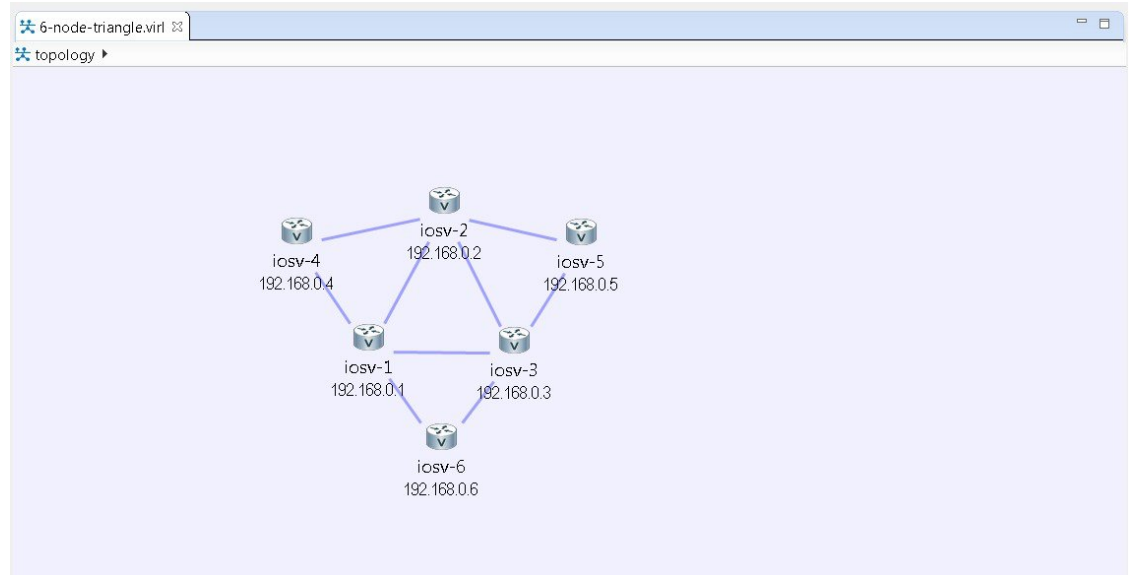
The following figure shows an overview of the AutoNetkit visualization phase as it appears in a browser window.

Figure 93: Visualization Overview



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

Figure 94: Topology Design

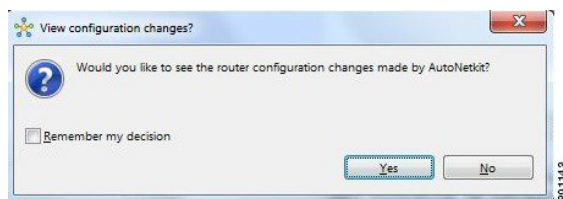


Access AutoNetkit Visualization

To access AutoNetkit visualization, complete the following steps:

-
- Step 1** Generate a configuration for the topology.
Click **Build Initial Configurations** from the toolbar. Alternatively, from the menu bar, choose **Configuration > Build Initial Configurations**.
- Step 2** View the configuration changes.
AutoNetkit displays a notification after it generates the configuration.

Figure 95: View Configuration Change Notification

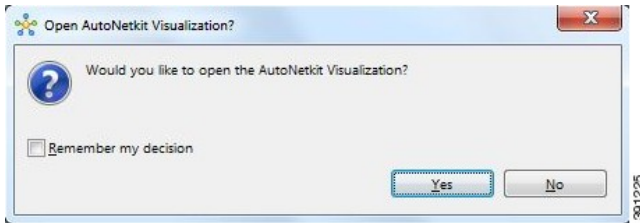


- Click **No** to skip this step.
- Click **Yes** to open a comparison view of the configuration changes.

- Step 3** Display the AutoNetkit visualization view of the topology.

When you close the comparison view, a notification prompts you whether to open AutoNetkit visualization.

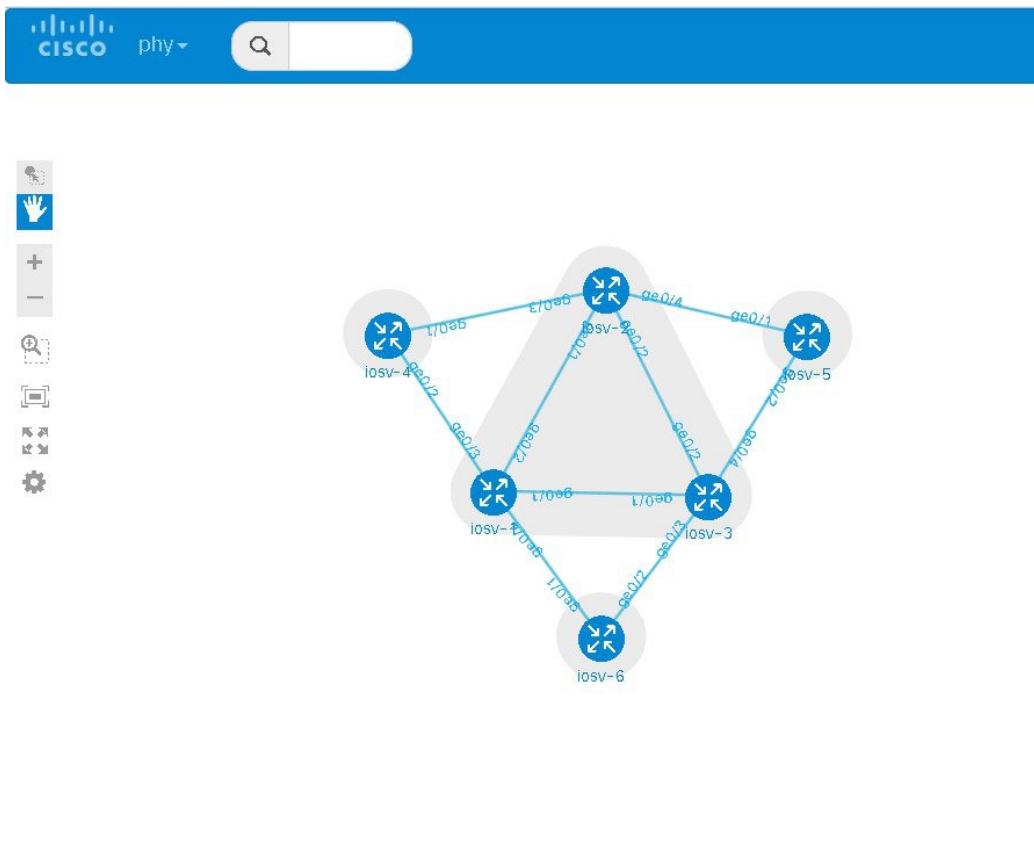
Figure 96: Open AutoNetkit Visualization



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

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

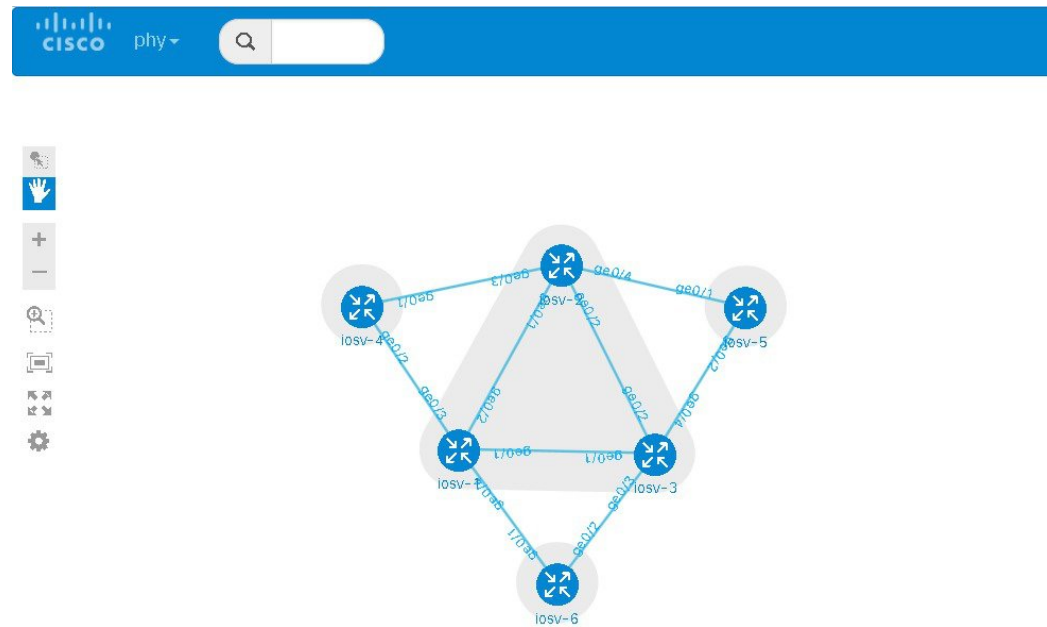
Figure 97: AutoNetkit Visualization Window



AutoNetkit View Options

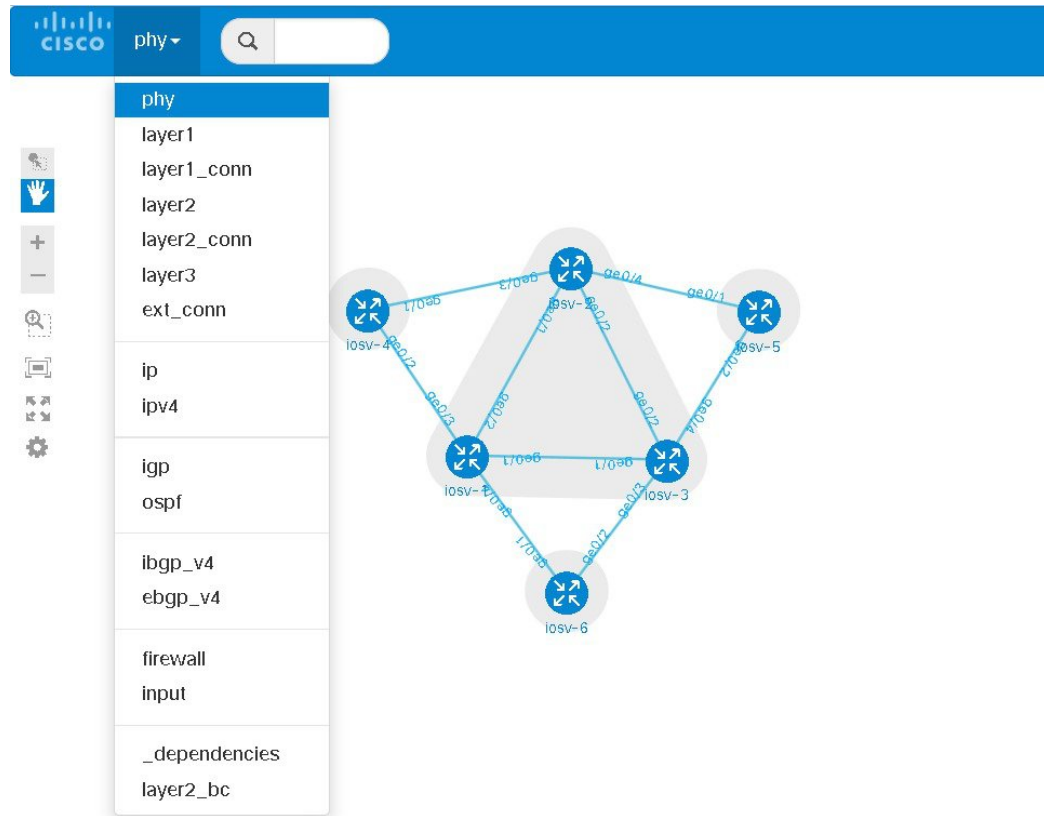
The initial AutoNetkit visualization view that is displayed in the browser window is the physical model of the topology. The physical model shows the nodes and interface connections between the nodes. It is similar to the Cisco Modeling Labs topology view.

Figure 98: Initial View



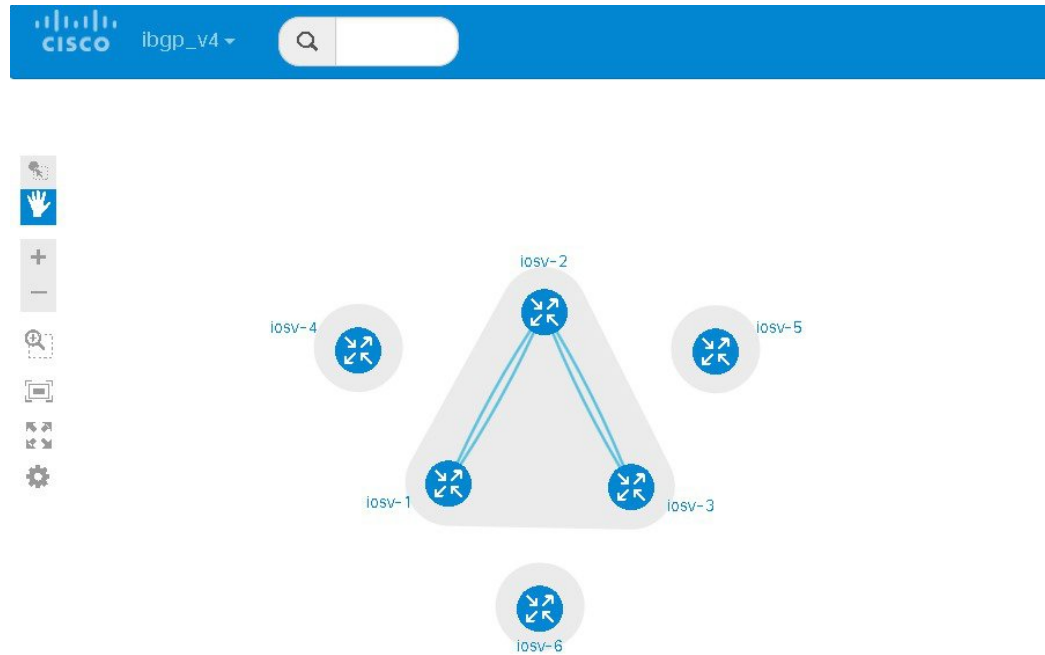
To select another view, place the cursor over the **phy** selection in the browser window.

Figure 99: List of Available Views



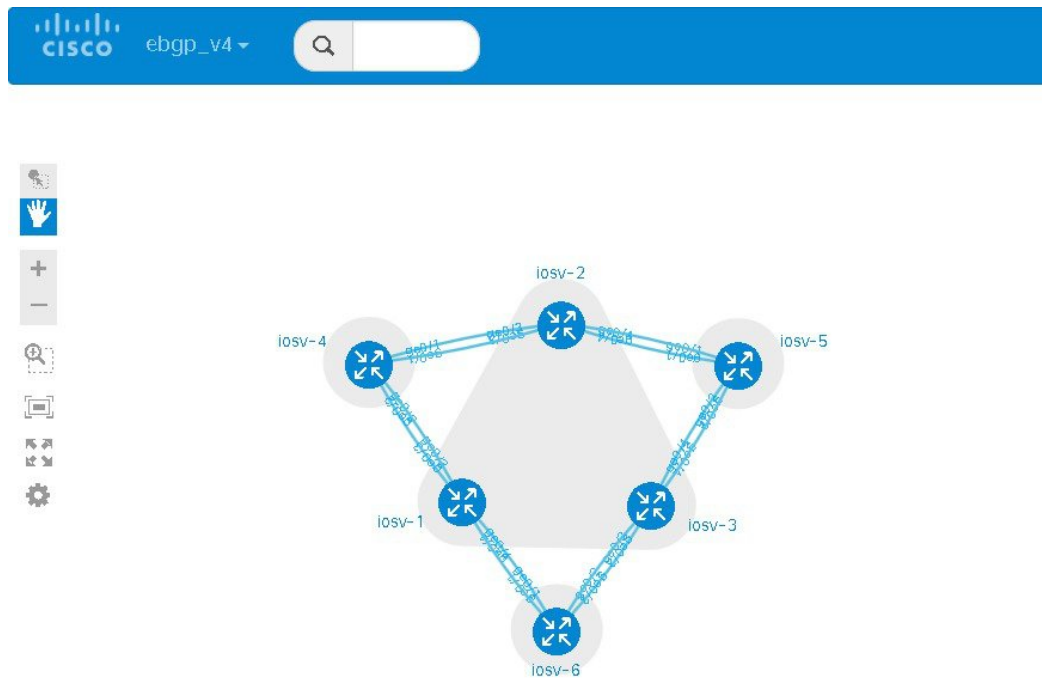
When you place the cursor over the Overlay view, several choices appear. For example, selecting **ibgp_v4** will show the IPv4 iBGP topology.

Figure 100: ibgp_v4 View



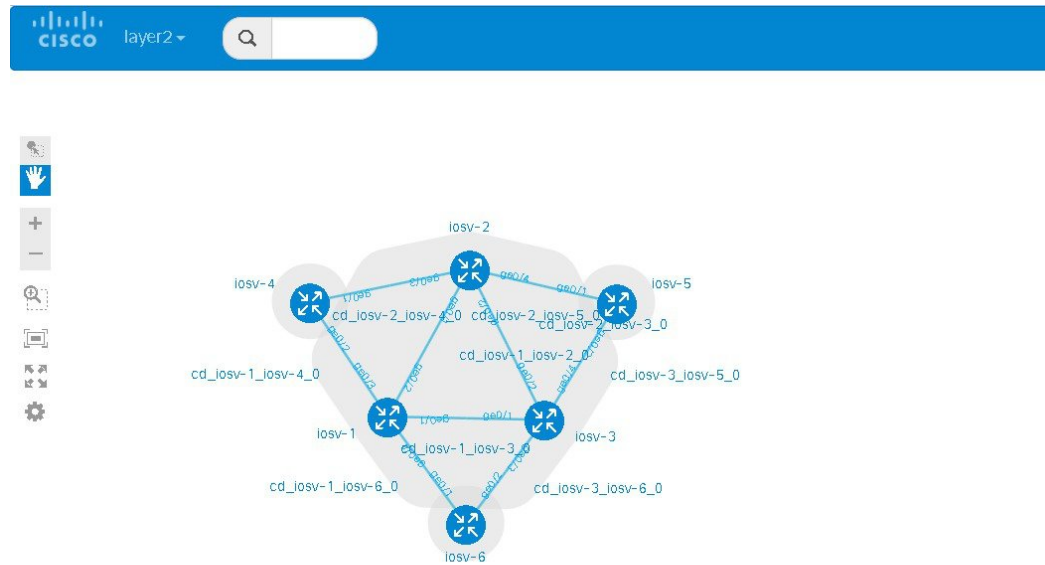
For example, selecting **ebgp_v4** will show the IPv4 eBGP topology.

Figure 101: ebgp_v4 View



For example, selecting **layer2** will show the IPv4 Layer 2 topology.

Figure 102: Layer 2 View

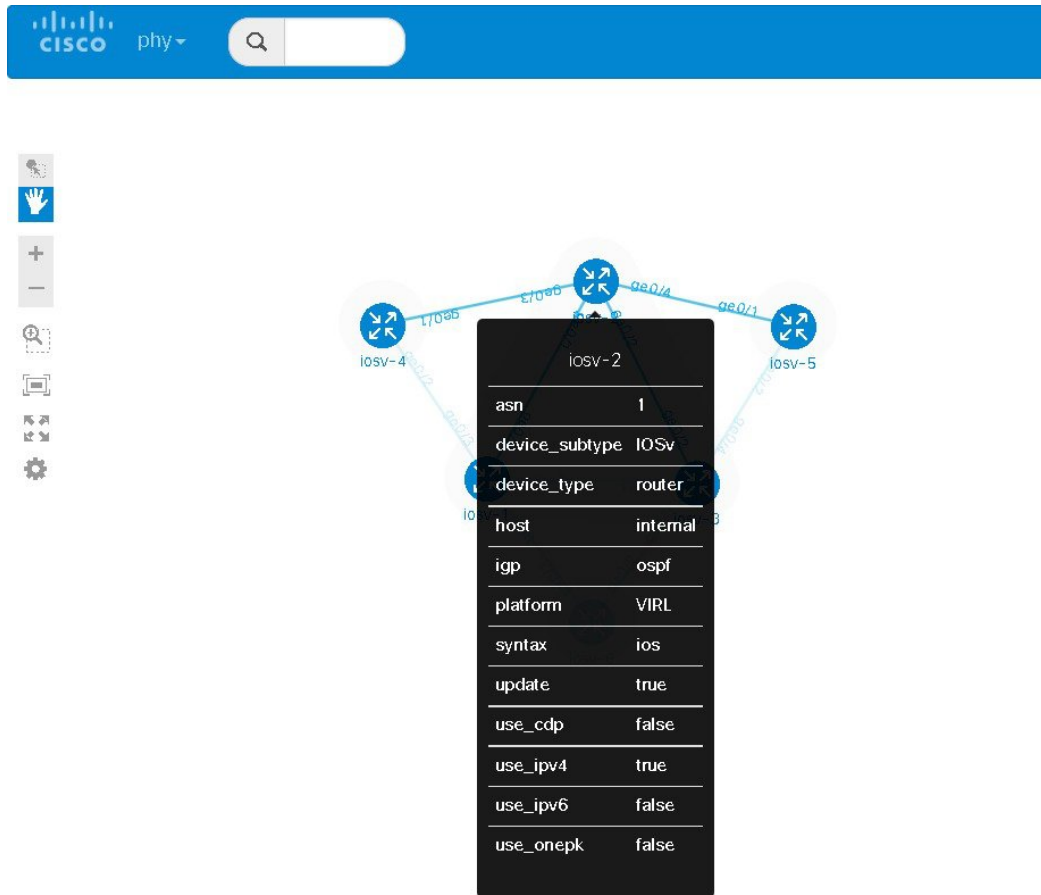


Placing the cursor over one of the nodes displays a pop-up view of information about that node. The type of information displayed depends on the selected option and node configuration.

**Note**

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

Figure 103: Node Information



You can continue to select different protocol views to see how the protocol-centric view changes. In a complex topology, you can use the **phy** views to verify that the protocols, nodes, and connections meet the design requirements.



CHAPTER 6

Simulate the Topology

- [Simulate the Topology Overview, page 131](#)
- [Launch a Simulation, page 132](#)
- [Connect to a Simulation Node Console, page 148](#)
- [Start a Single Node, page 154](#)
- [Stop a Simulation, page 160](#)
- [Stop a Single Node, page 164](#)
- [Modify a Node Configuration in the Simulation, page 167](#)
- [Extract and Save Modified Configurations, page 172](#)
- [Linux Server Snapshot Support, page 175](#)

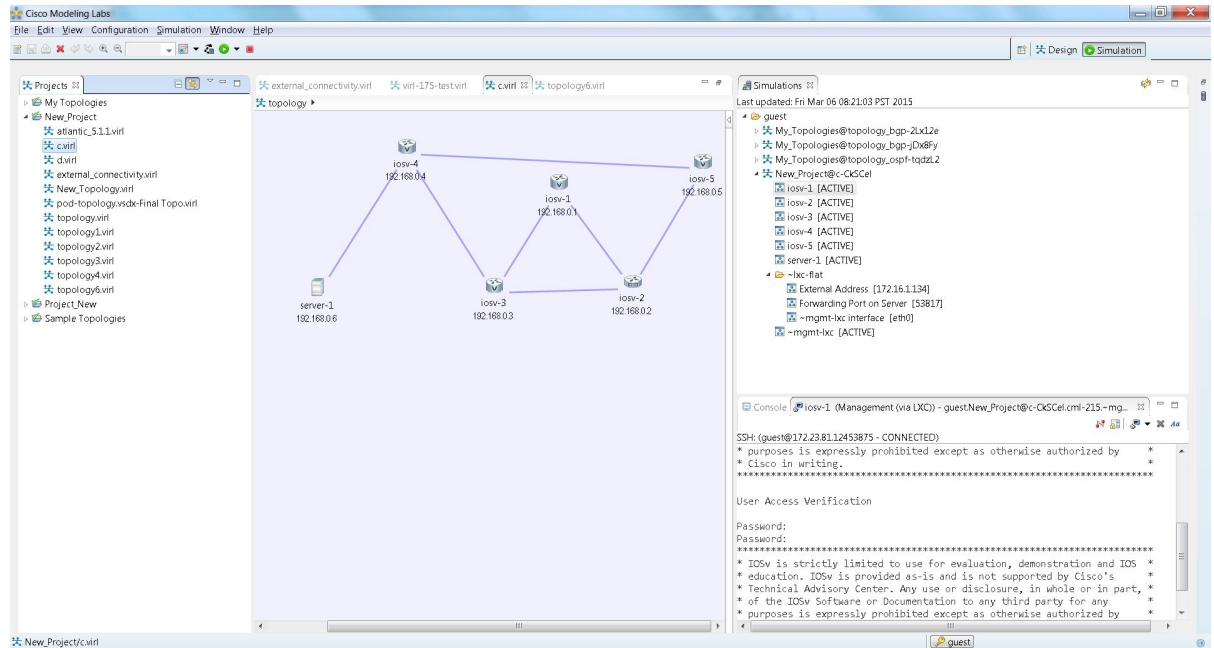
Simulate the Topology Overview

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 **Simulations** and **Console** views for a running simulation.

Figure 104: Simulation Perspective Initial Display



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 [Customize Perspectives](#), on page 22 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, the number of nodes you can run is limited by the resources available on the server and the number of licenses available. Currently, the design limit is 1,000 nodes.

Launch a Simulation

To launch a simulation, complete the following steps.

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 autogeneration, 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.

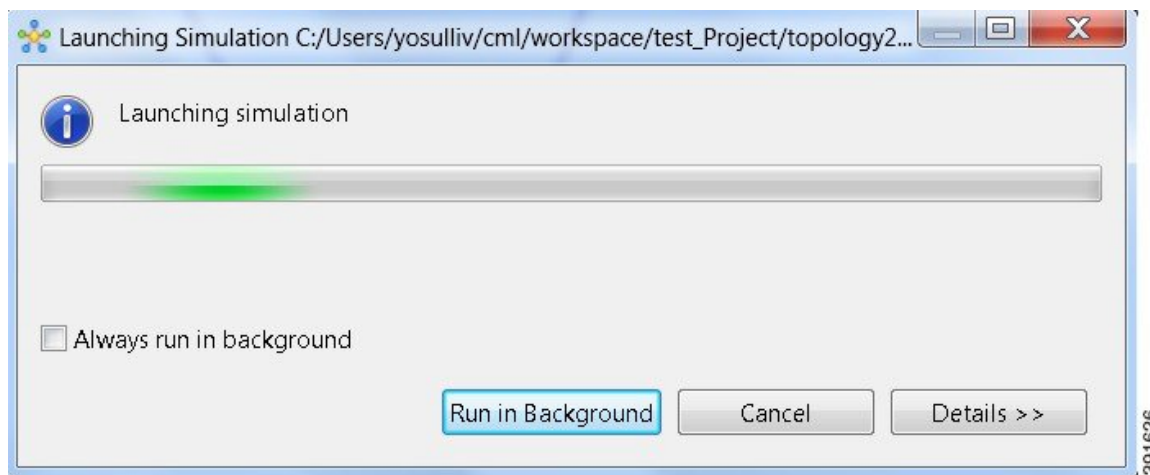
Step 1

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:



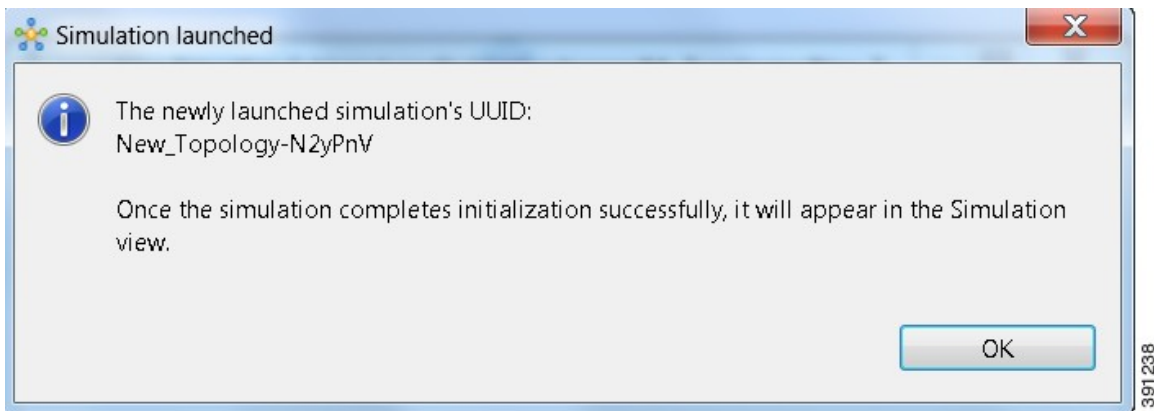
- a) Check the **Always run in the background** check box. All future node start requests, stop requests, and simulation launch requests run in the background and do not display dialog boxes.

Note To control the background setting, choose **File > Preferences > General**.

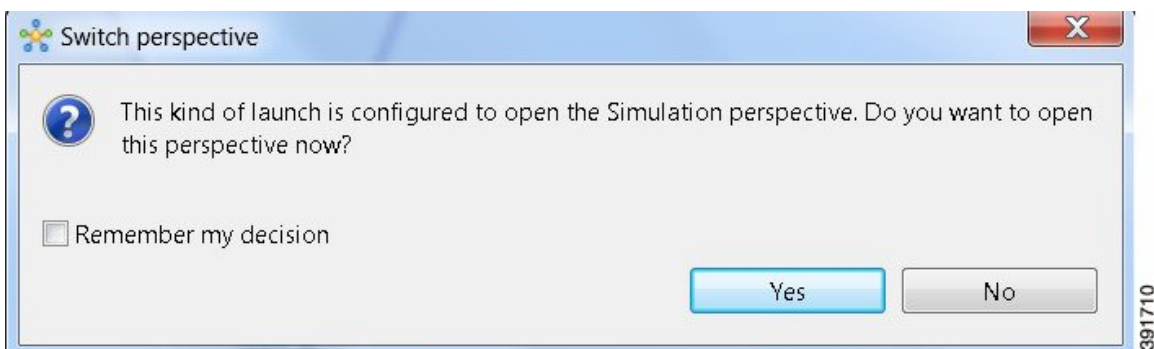
- b) Click **Run in Background**. The dialog box closes when the node simulation starts.
- c) Click **Cancel** to return to the **Design** view.
- d) Take no action, and the node simulation launches momentarily.

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**.

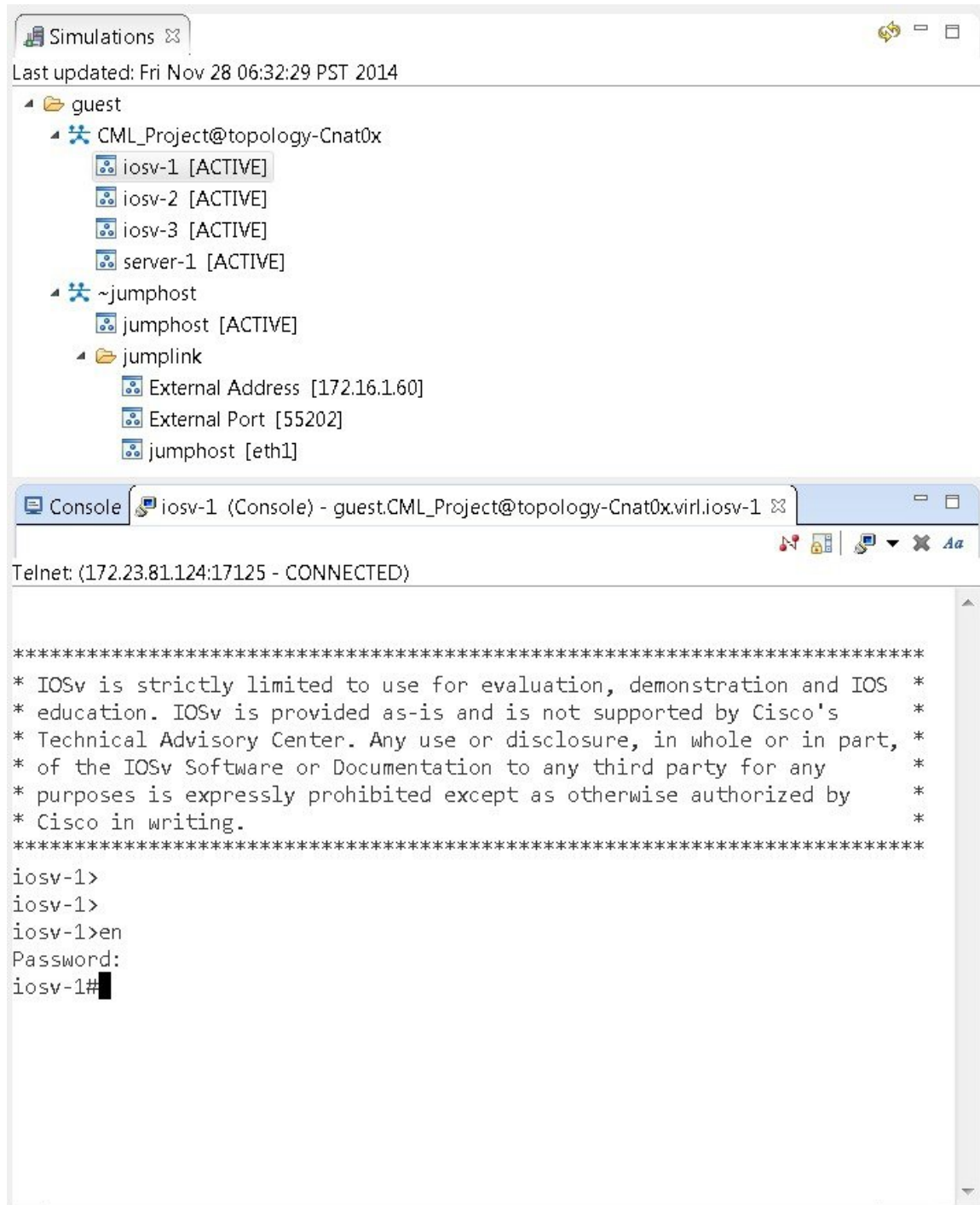


Note We recommend that you switch to the **Simulation** perspective to view the running simulation and to connect to node consoles.

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 **[ACTIVE]**, as shown in the following figure.

Figure 105: Simulation Launched in Simulations View



Jumphost Virtual Machine (VM)

The jumphost VM is the default method for accessing the management network of a running simulation. The jumphost node runs in a separate simulation named `~jumphost` and has two interfaces, `eth0` in the project/user management network and `eth1` in the FLAT network.

The purpose of the jumphost is to provide an access point into a simulated network that remains fixed, in that there is a single external IP address or port that the user can access. A user can access the jumphost and then access all the nodes inside the simulation.

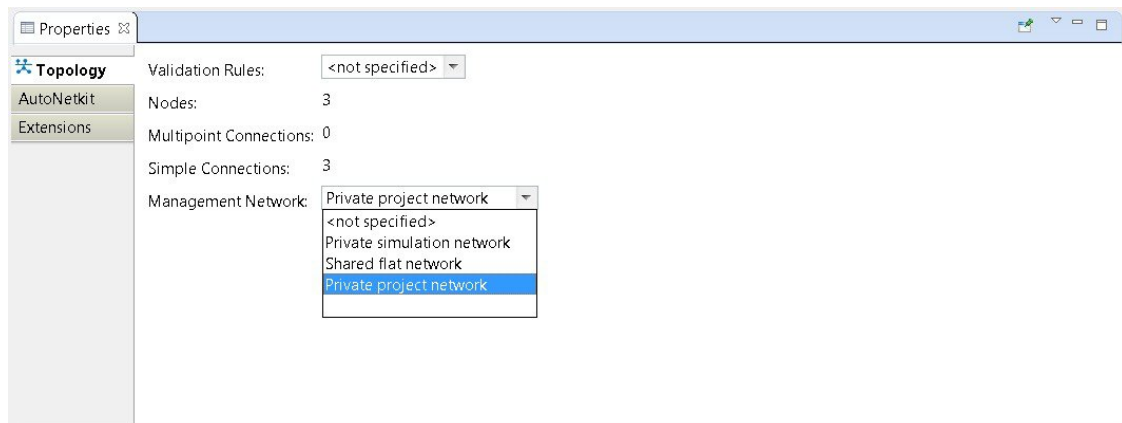
Cisco Modeling Labs provides two implementations of jumphost:

- A VM: Based on the server VM image type.
- A Linux container: A lighter weight form of a jumphost. See [Linux Container \(LXC\)](#), on page 137 for more information.

The VM implementation is costly in terms of the memory and CPU used when a jumphost is instantiated. However, since it is a full-blown server VM, there is value to it, in that you can install and run any application on it.

To select a jumphost VM, in the Cisco Modeling Labs client, choose **Properties > Topology > Management Network > Private project network** or **Shared flat network**.

Figure 106: Setting the Jumphost Option



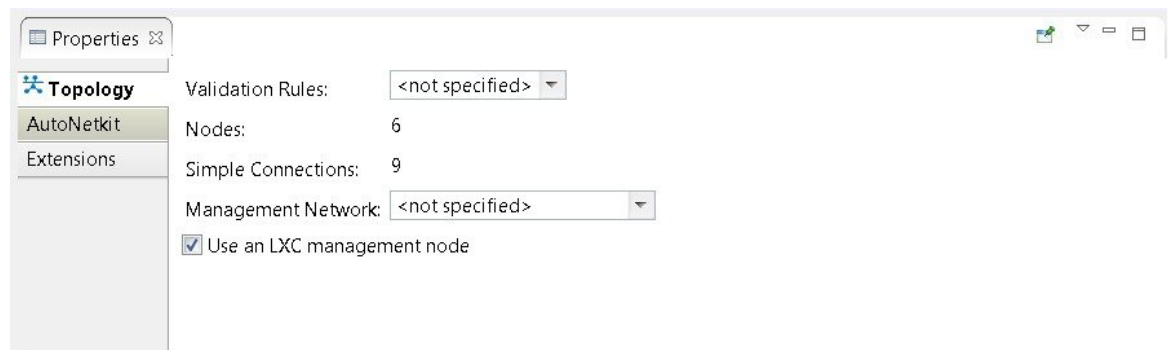
The jumphost VM is per user; a user can create multiple simulations, but only one corresponding jumphost is created. Therefore, depending on the type of simulation you are running, you can choose between the two implementations.

Linux Container (LXC)

An LXC provides a means of accessing a topology rather than having to create a full Linux server VM. All nodes in the network are connected to a hidden OOB management network that uses the first interface on each of the nodes.

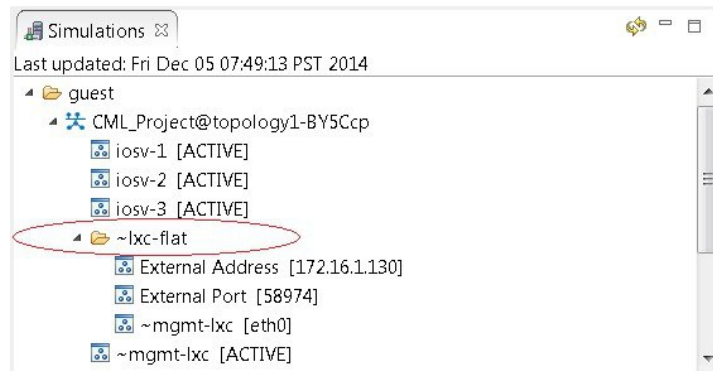
In the Cisco Modeling Labs client, it is enabled by selecting the management network type **Private simulation network** for your topology. The LXC is automatically connected into this hidden OOB management network to which all VMs in your simulation are connected. This enables you to connect into each VM via its management Ethernet port, removing the need to use the console port connection method.

Figure 107: Setting the LXC Option



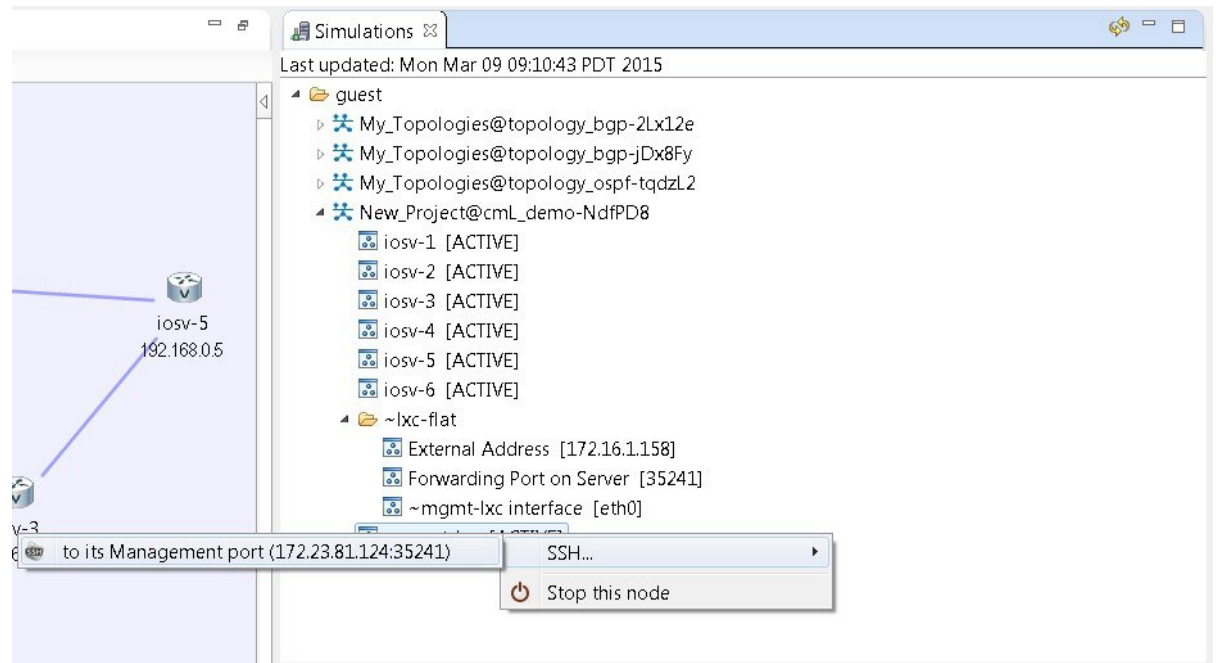
The LXC is operational when your simulation is active and terminates when your simulation stops. LXC uses a significantly smaller memory footprint than the Linux server VM. The LXC automatically gets an IP address on the FLAT network.

Figure 108: LXC on the FLAT Network



LXC facilitates SSH access to the VMs; it provides direct SSH access to each VM running inside the simulation. Telnet is not supported. As shown below, LXC is accessed by right-clicking the `~mgmt_lxc [ACTIVE]` node and selecting **SSH** from the list.

Figure 109: Accessing LXC



The LXC is automatically spun up and provides a jump host point for access into the network. Connecting to the LXC means you can see the interfaces to the outside world and to the OOB network inside your simulation.

A connection is opened from the Cisco Modeling Labs client to a port on the Cisco Modeling Labs server and is forwarded to the LXC. The LXC, in turn, opens a connection to the Management Interface (Gi0/0) of the VM inside the simulation to the node instance.

Static Port Assignment to the LXC

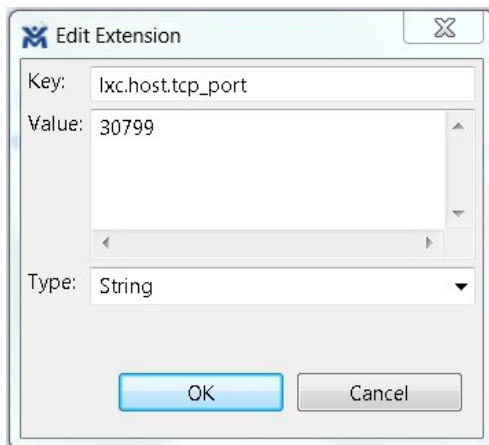
When the management network property **Private simulation network** is set, Cisco Modeling Labs assigns a random port for SSH port access to the LXC. However, you can statically define this by setting an extension on your topology.

To set an extension for your topology, complete the following steps.

-
- Step 1** Click on the canvas to open the **Topology** tab in the **Properties** view.
 - Step 2** Click the **Extensions** tab.
 - Step 3** Click the **Add new extension** icon.
The **Edit Extension** dialog box appears.
 - Step 4** Enter the following values:

- **Key:** `lxc.host.tcp_port`
- **Value:** `30799`
- **Type:** `String` (from the drop-down list)

Figure 110: Add a New Extension



- Step 5** Click **OK** to add the new extension.
When the LXC starts, it will be bound to the TCP port specified in the new extension.
-

Launch a Phased Simulation

On occasions, you may need to start your simulation in phases rather than having all nodes launched at the same time. This functionality is facilitated by the **Exclude node from simulation launch** check box, which allows you to pick and choose which nodes to start.

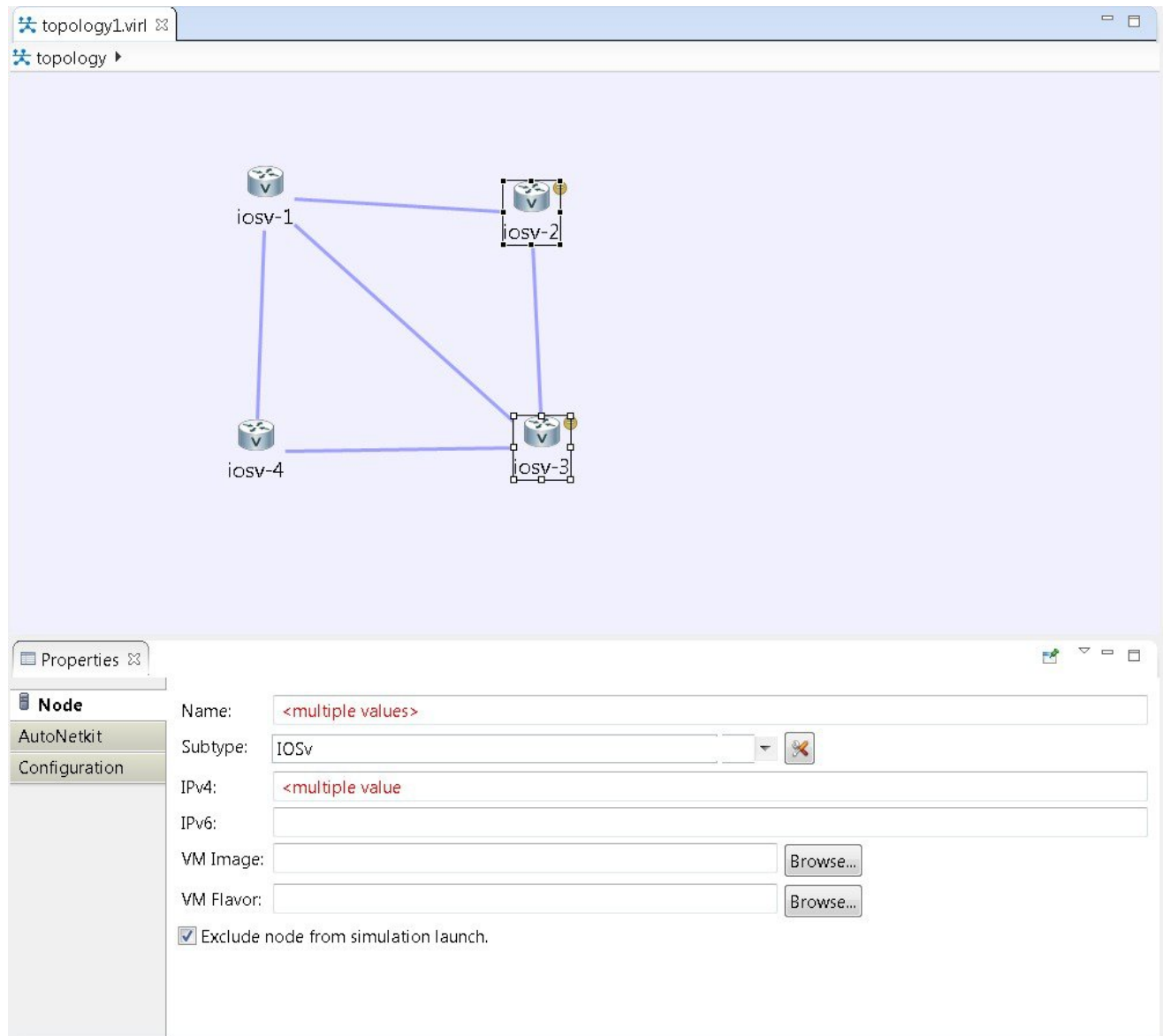
To launch a phased simulation, complete the following steps.

- Step 1** With the applicable topology open, click on the canvas and move the selection area over the nodes to be excluded from the running simulation. (Alternatively, you can double-click a specific node, hold down the **Shift** key, and select the remaining nodes.)

The **Properties > Node** view opens.

Step 2 In the **Properties > Node** view, check the **Exclude node from simulation launch** check box.

Figure 111: Nodes Excluded from Simulation Start



Step 3 Save your topology using **Ctrl-S**.

Note You can set this property for each individual node, if you prefer. Select the node on the canvas, and select the **Exclude node from simulation launch** check box in the **Properties > Node** view.

Step 4 From the toolbar, click the **Build Initial Configurations** button to build the node configurations.

Step 5 From the toolbar, click the **Launch Simulation** button.
The simulation launches.

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

Note In the **Console** view, you can see the message **Node '<node name>' is excluded from automatic start** for the excluded nodes, and in the **Simulations** view, you can see that the excluded nodes have the state **[ABSENT]**.

Figure 112: Phased Simulation Launched

Simulations ✕

Last updated: Sat Aug 15 12:37:18 PDT 2015

- iosv-1 [ACTIVE]
 - iosv-2 [ABSENT]
 - iosv-3 [ABSENT]
 - iosv-4 [ACTIVE]
 - ~lxc-flat
 - External Address [172.16.1.75]
 - Forwarding Port on Server [45069]
 - ~mgmt-lxc interface [eth0]
 - ~mgmt-lxc [ACTIVE]

Console ✕

```

topology1.virl simulation New_Project@topology1-RGXfvd
(INFO) [Aug/15/2015 19:36:41] Node "iosv-2" is excluded from automatic start
(INFO) [Aug/15/2015 19:36:42] Node "iosv-3" is excluded from automatic start
(INFO) [Aug/15/2015 19:36:42] Session "New_Project@topology1-RGXfvd": nodes can start
(INFO) [Aug/15/2015 19:36:45] Starting node "iosv-4"
(INFO) [Aug/15/2015 19:36:45] Starting node "iosv-1"
(INFO) [Aug/15/2015 19:36:51] Starting node "~mgmt-lxc"
(INFO) [Aug/15/2015 19:36:59] Node "iosv-1" state changed from ABSENT to BUILDING
(INFO) [Aug/15/2015 19:36:59] Node "iosv-4" state changed from ABSENT to BUILDING
(INFO) [Aug/15/2015 19:37:16] Node "iosv-1" state changed from BUILDING to ACTIVE
(INFO) [Aug/15/2015 19:37:16] Node "iosv-4" state changed from BUILDING to ACTIVE
  
```

406004

Launch a Time-Limited Simulation

In circumstances where you need to run a simulation for a specified time, complete the following steps.

Step 1 From the toolbar, click the down arrow to the right of the **Launch Simulation** button.

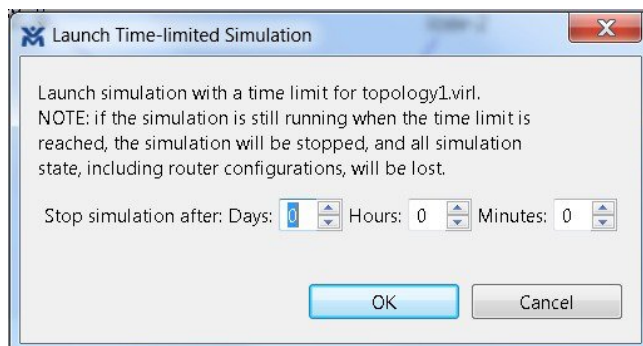
Step 2 Click **Launch Simulation (with time limit)**.

Figure 113: Launch Simulation with Time Limit Option



The **Launch Time-limited Simulation** dialog box is displayed.

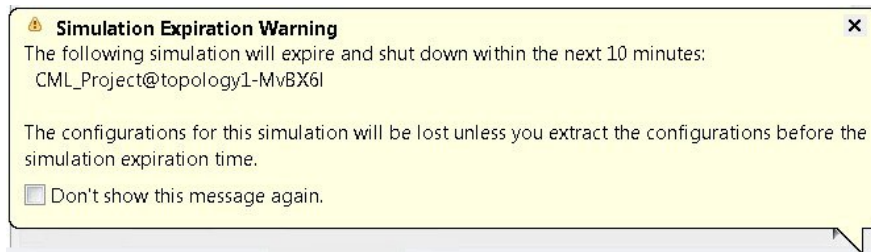
Figure 114: Launch a Time-Limited Simulation



Step 3 Set a time duration for the simulation by entering details for **Days**, **Hours**, and **Minutes** by using either the up and down arrows or entering the values directly. Click **OK** to apply your time limit to the simulation. The simulation launches.

Note An expiration warning is displayed ten minutes (or less) before your simulation is due to expire.

Figure 115: Simulation Expiration Warning



Note We recommend that you extract configurations for your simulation before it expires. You can reset the time limit for a running simulation in the **User Workspace Management** interface. See [Reset the Time Limit on a Running Simulation](#), on page 144 for more information.

Reset the Time Limit on a Running Simulation

You can extend or reduce the time limit set for a running simulation in the **User Workspace Management** interface. To do this, complete the following steps.

- Step 1** Log in to the **User Workspace Management** interface. On the **Overview** page, information on running simulations is displayed. Move to the applicable simulation under the **Session** heading. If your simulation is due to expire in ten minutes or less, the simulation name is displayed in red.
- Step 2** Under the **Options** column, click the down arrow and click **Set expiration**.

Figure 116: Soon to Expire Simulation

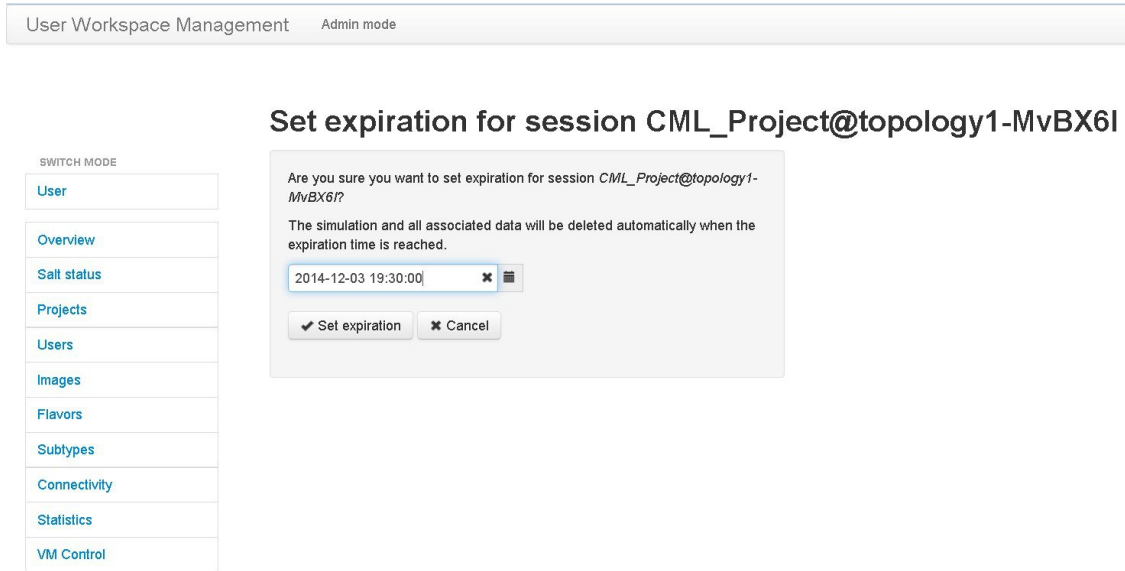
Sessions Request to stop all

Sessions of project *guest* Request to stop selected

Session	User	Project	Status	Expires	Options
<input type="checkbox"/> CML_Project@topology1-MvBX6l	guest	guest	ACTIVE	2014-12-03 19:18:00 (in 0d 00h 09m)	<input type="checkbox"/> Request stop <input checked="" type="checkbox"/> Set expiration
<input type="checkbox"/> ~jumphost	guest	guest	ACTIVE	never	<input type="checkbox"/> Request stop <input type="checkbox"/> Set expiration

Step 3 The **Set expiration for session** page is displayed. In the date and time field, enter either a new expiration date and time, a date only, or a time only for the simulation.

Figure 117: Set Expiration for Session Page



Step 4 Click **Set expiration** to save the changes. The time limit for the simulation is updated.

Control Interface States

In a running simulation, you are able to change the state of the network interface by bringing it up or down.



Note This changes the state of the underlying communication infrastructure, not the interface state of the virtual machine.

To control the state of an interface, complete the following steps.

- Step 1** Log in to the **User Workspace Management** interface.
- Note** You must log in as a user other than the uwadmin user, for example, guest.
- Step 2** On the **Overview** page, under **Sessions**, choose the applicable running session.

A list of active virtual machines and interfaces is displayed.

Step 3 Scroll down to the **Interfaces** section and choose the applicable virtual machine.

Step 4 From the applicable **Options** drop-down list, click **Update admin state**.

Figure 118: Interface State Control Option

Interfaces

Node	Interface name	Interface up	Network subtype	Network name	IP Addresses	Options
iosv-1	GigabitEthernet0/0	True	SESSION MGMT	mgmt	10.255.0.5 / 16	▼
iosv-1	GigabitEthernet0/1	True	SIMPLE	iosv-1-to-iosv-3	unassigned	▼
iosv-1	GigabitEthernet0/2	True	SIMPLE	iosv-1-to-iosv-2	unassigned	Update admin state
iosv-1	GigabitEthernet0/3	True	SIMPLE	iosv-1-to-iosv-4	unassigned	▼
iosv-1	GigabitEthernet0/4	True	SIMPLE	iosv-1-to-iosv-6	unassigned	▼
iosv-2	GigabitEthernet0/0	True	SESSION MGMT	mgmt	10.255.0.6 / 16	▼
iosv-2	GigabitEthernet0/1	True	SIMPLE	iosv-1-to-iosv-2	unassigned	▼
iosv-2	GigabitEthernet0/2	True	SIMPLE	iosv-2-to-iosv-3	unassigned	▼
iosv-2	GigabitEthernet0/3	True	SIMPLE	iosv-2-to-iosv-4	unassigned	▼
iosv-2	GigabitEthernet0/4	True	SIMPLE	iosv-2-to-iosv-5	unassigned	▼
iosv-3	GigabitEthernet0/0	True	SESSION MGMT	mgmt	10.255.0.7 / 16	▼

Depending on the current state of the interface, either a **Bring Down** or **Bring Up** page is displayed. In this case, the **Bring Down** page is displayed.

Figure 119: Bring Down the Applicable Interface

Bring down interface GigabitEthernet0/1 on node iosv-1

Are you sure you want to bring down interface **GigabitEthernet0/1** on node **iosv-1**?

Step 5 Click **Bring down** to bring down the network interface.

A message is displayed indicating that the interface has been brought down.

Figure 120: Interface Successfully Brought Down



Session Sample_Topologies@6-node-triangle-WPUM0y details

Nodes

Node	Subtype	State	Options
iosv-1	IOSv	ACTIVE	▼
iosv-2	IOSv	ACTIVE	▼
iosv-3	IOSv	ACTIVE	▼
iosv-4	IOSv	ACTIVE	▼
iosv-5	IOSv	ACTIVE	▼
iosv-6	IOSv	ACTIVE	▼
~mgmt-lxc	mgmt-lxc	ACTIVE	

406558

Step 6 To bring up the interface at a later stage, click **Update admin state** again. The **Bring up** page displayed.

Figure 121: Bring Up the Applicable Interface

Bring up interface GigabitEthernet0/1 on node iosv-1



406559

Step 7 Click **Bring up** to bring up the network interface.

A message is displayed indicating that the interface has been brought up.

Figure 122: Interface Successfully Brought Up

Interface GigabitEthernet0/1 on node iosv-1 successfully brought up

Session Sample_Topologies@6-node-triangle-WPUM0y details

Nodes

Node	Subtype	State	Options
iosv-1	IOSv	ACTIVE	▼
iosv-2	IOSv	ACTIVE	▼
iosv-3	IOSv	ACTIVE	▼
iosv-4	IOSv	ACTIVE	▼
iosv-5	IOSv	ACTIVE	▼
iosv-6	IOSv	ACTIVE	▼
~mgmt-kc	mgmt-kc	ACTIVE	

406560

Connect to a Simulation Node Console

Cisco Modeling Labs provides the capability for you to connect to your nodes via SSH and Telnet. You can start an SSH session, which connects into the node via the LXC, as described in [Linux Container \(LXC\)](#), on page 137.

This access method makes use of SSH to the LXC and then Telnet from the LXC to the nodes running inside the simulation. This does not use the console port of the nodes and is more reliable and faster to use.

Connect to a Simulation Node Console via SSH

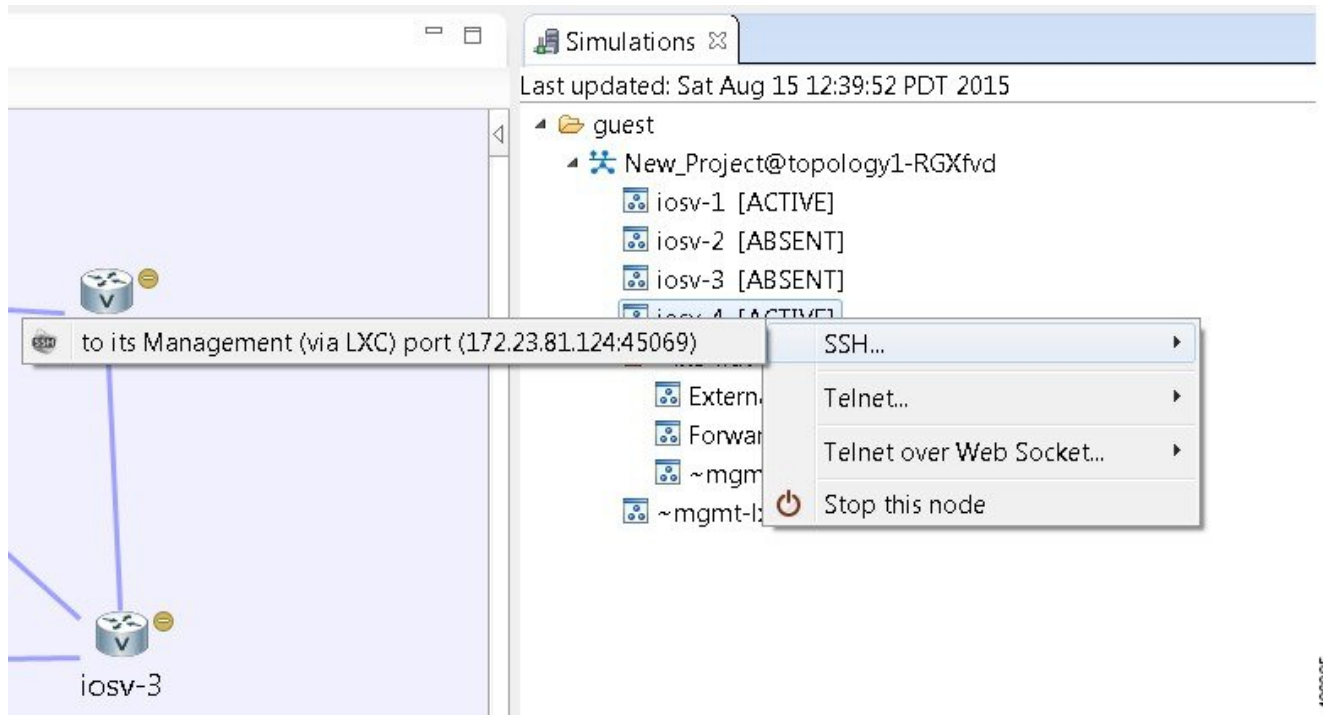
To connect to a simulation node console, complete the following steps.

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 **SSH > to its Management (via LXC) port**.

Figure 123: Connecting to a Node Console via SSH



406095

A new **Terminal** view opens.

Figure 124: Terminal View

```

Console | iosv-4 (Management (via LXC)) - guest.New_Project@topology1-RGXfvd.virt~mgmt-lxc
SSH: (guest@172.23.81.12445127 - CONNECTED)
guest@mgmt-RGXfvd$
guest@mgmt-RGXfvd$
guest@mgmt-RGXfvd$ telnet 10.255.0.110
Trying 10.255.0.110...
Connected to 10.255.0.110.
Escape character is '^]'.

*****
* IOSv is strictly limited to use for evaluation, demonstration and IOS *
* education. IOSv is provided as-is and is not supported by Cisco's *
* Technical Advisory Center. Any use or disclosure, in whole or in part, *
* of the IOSv Software or Documentation to any third party for any *
* purposes is expressly prohibited except as otherwise authorized by *
* Cisco in writing. *
*****

User Access Verification

Password:
*****
* IOSv is strictly limited to use for evaluation, demonstration and IOS *
* education. IOSv is provided as-is and is not supported by Cisco's *
* Technical Advisory Center. Any use or disclosure, in whole or in part, *
* of the IOSv Software or Documentation to any third party for any *
* purposes is expressly prohibited except as otherwise authorized by *
* Cisco in writing. *
*****

iosv-4>

```

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.

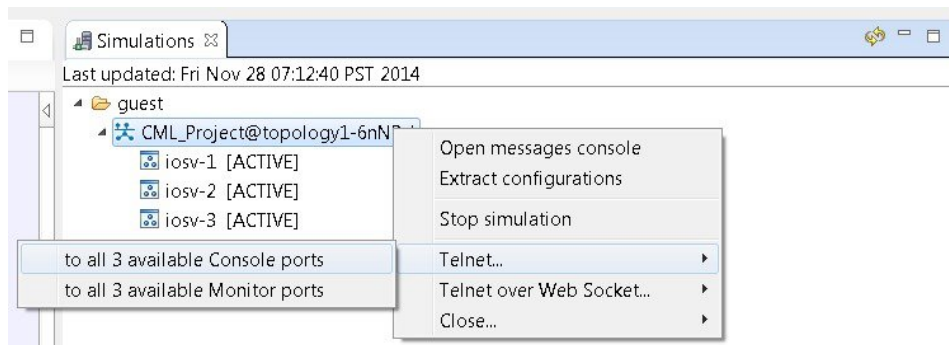
Tip When you disconnect a terminal but do not close the **Terminal** window, you can press **Enter** to reconnect the terminal.

Connect to Multiple Simulation Node Consoles

To connect to all consoles for all nodes in a running simulation, complete the following steps.

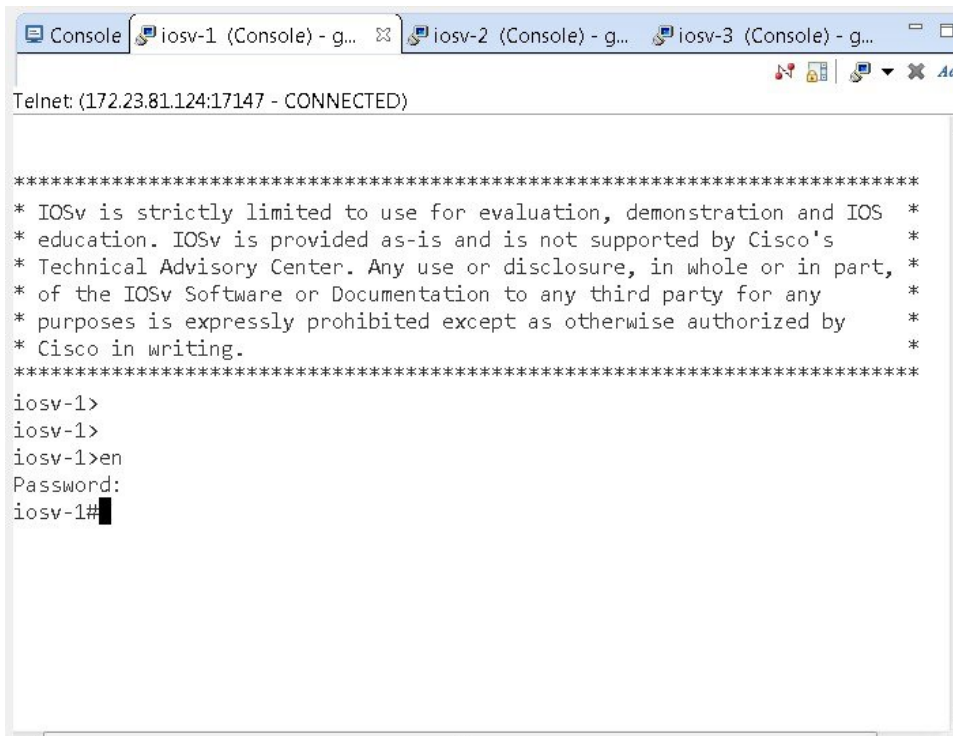
Step 1 Right-click the simulation in the **Simulations** view and choose **Telnet > to all <number> available Console ports**.

Figure 125: Connect to Multiple Node Consoles



A new **Terminal** view opens for all console ports.

Figure 126: 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 When you disconnect a terminal but do not close the **Terminal** window, you can press **Enter** to reconnect the terminal.

Terminal Multiplexer Functionality

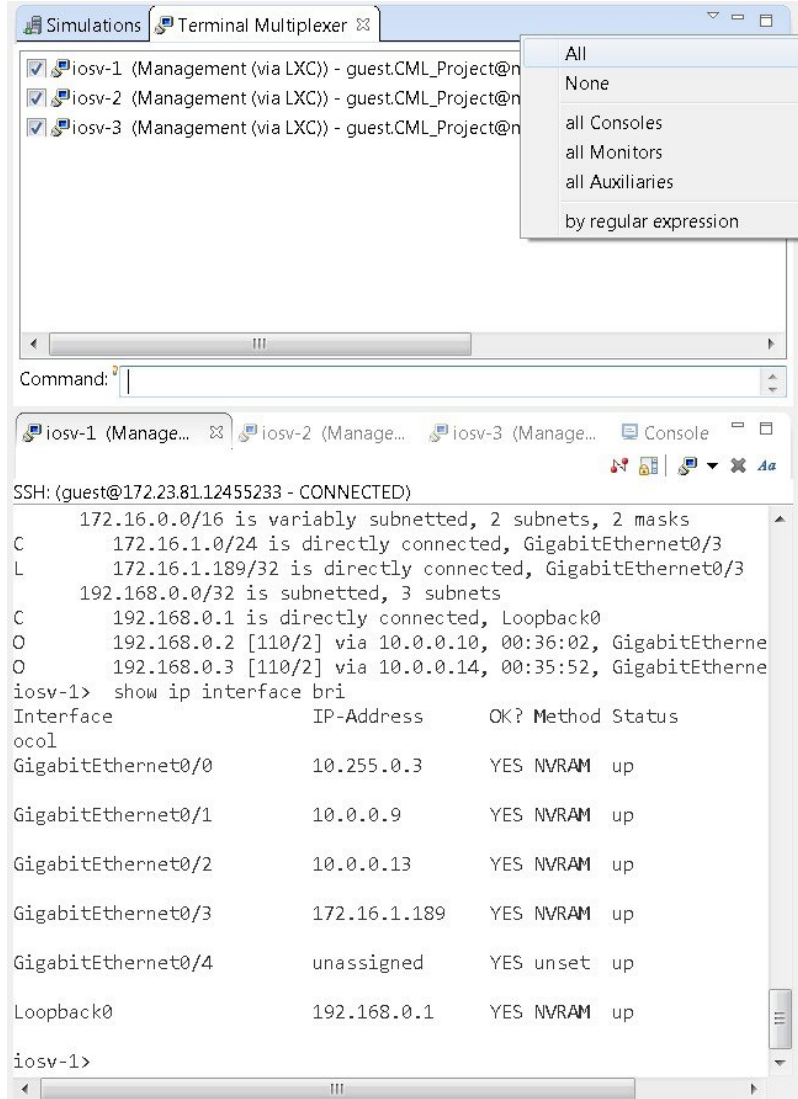
A terminal multiplexer is available for use with the Cisco Modeling Labs client. It permits a number of terminals to be accessed and controlled from a single terminal. Terminals can be detached to run in the background and then reattached later.

The terminal multiplexer is available from **Window > Show View > Other > General > Terminal Multiplexer**.

When the terminal multiplexer starts up, a status line at the bottom of the terminal displays information on the current session and is used to enter interactive commands. It lists all of the active console sessions. You

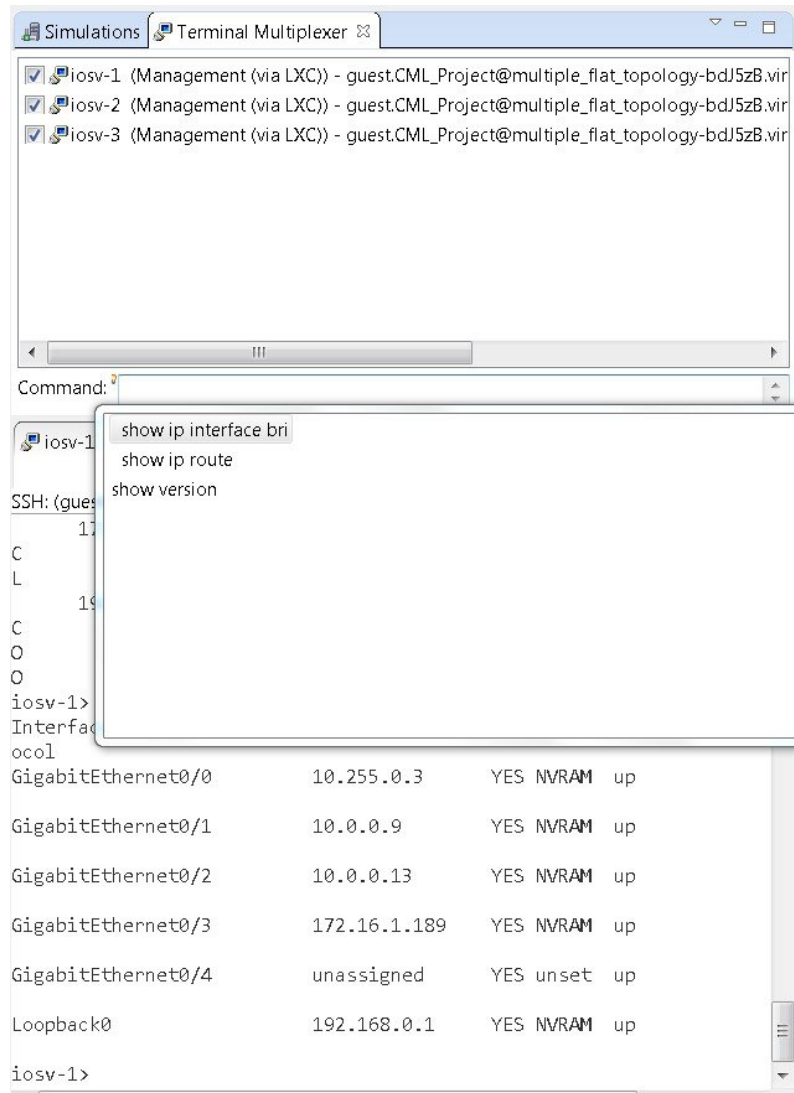
can select console sessions individually or all together from the **View Menu** drop-down list. Keyboard commands entered in the command-line text box are broadcast to the selected sessions.

Figure 127: Terminal Multiplexer



The terminal multiplexer also provides a command-line history, which you can access using **Ctrl-Space**.

Figure 128: Accessing the Command-Line History



Start a Single Node

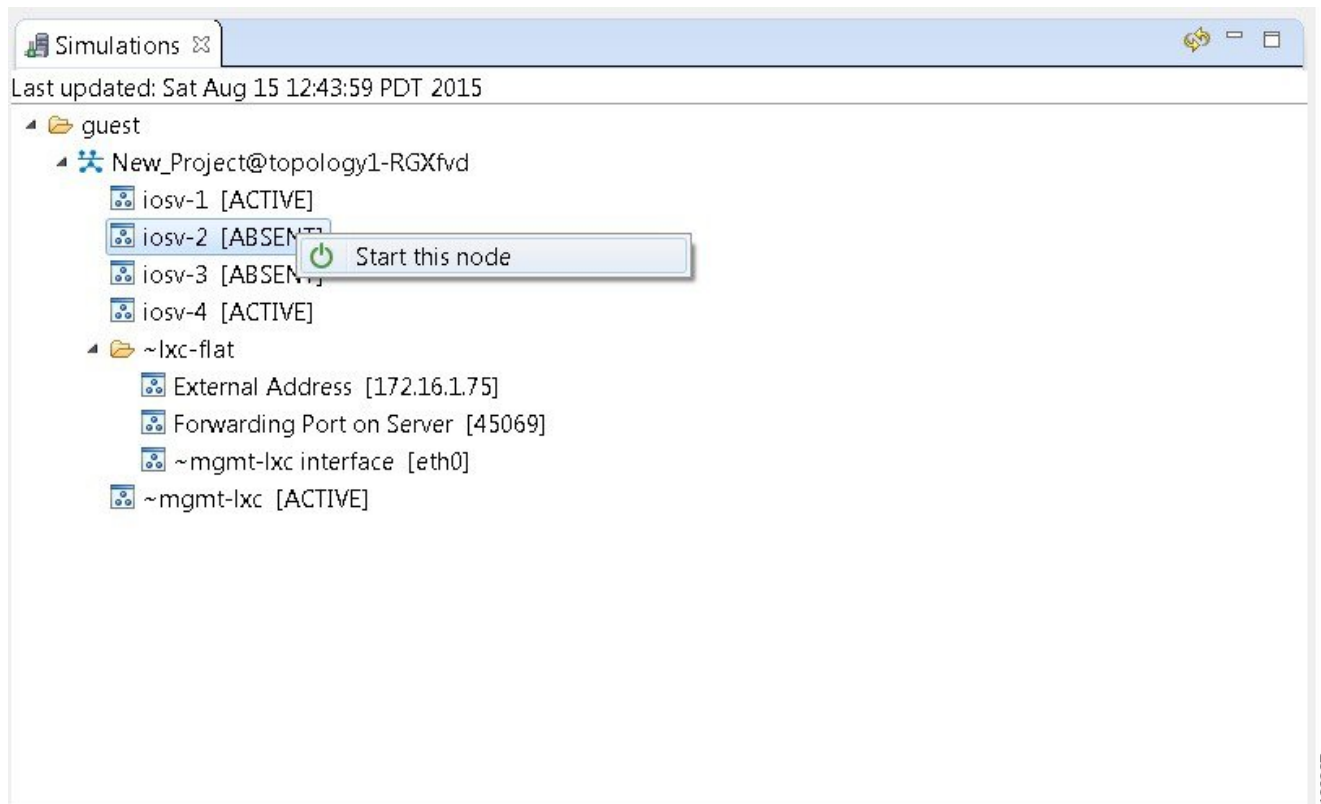
To start a single node, complete the following steps.

Step 1 Right-click a stopped node.

Step 2

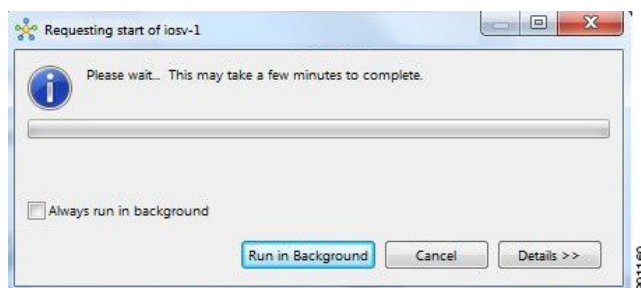
When a node is stopped, its status is shown as [ABSENT].
Click **Start this node**.

Figure 129: Start a Single Node



The **Requesting start** dialog box appears.

Figure 130: Requesting Start Dialog Box

**Step 3**

Choose one or more of the following actions:

- 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 displays a red **X**. Click the error icon to display the error dialog box.

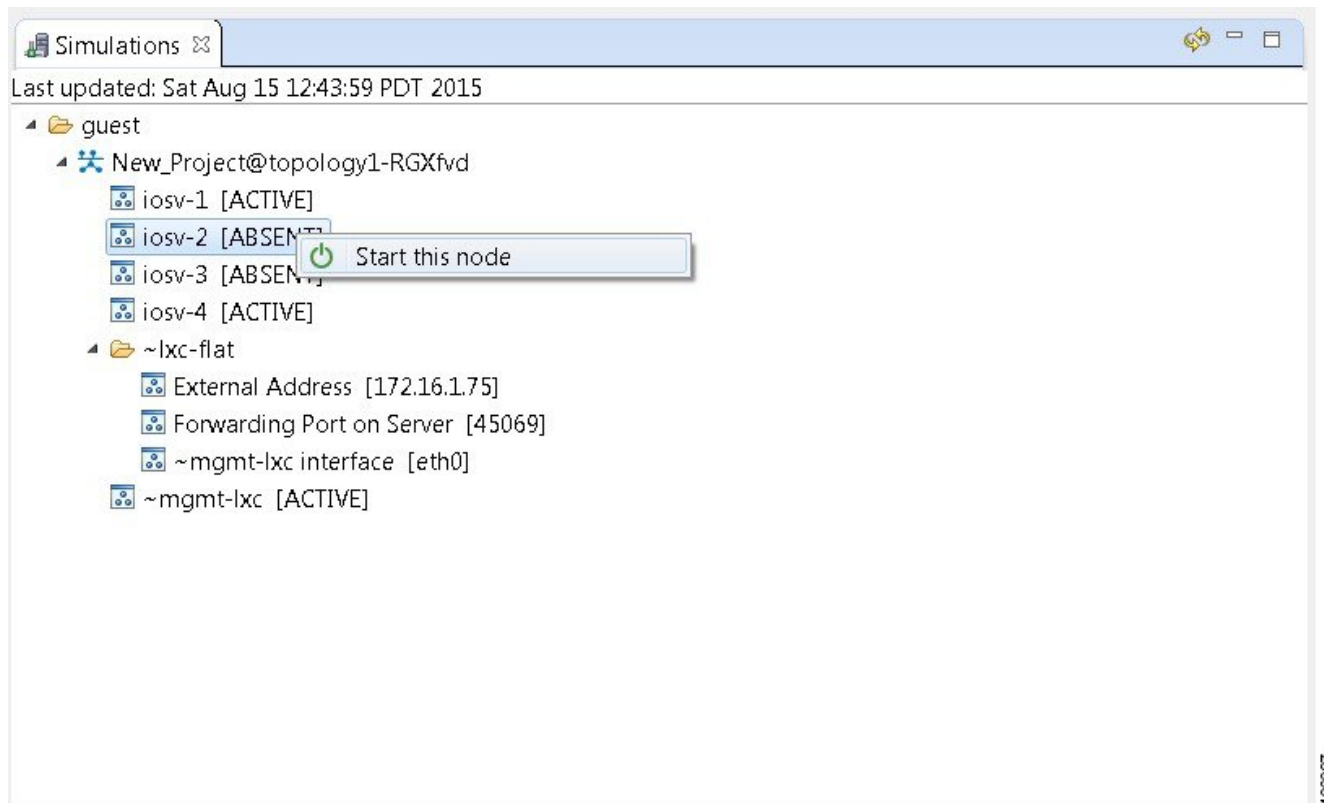
Start a Node in a Running Simulation

In cases where a phased simulation is running, you can later start those nodes not started with the initial simulation. To start a node in a running simulation, complete the following steps.

-
- Step 1** In the **Simulations** view, right-click the node.
- Note** A node not yet started has the status **[ABSENT]**.

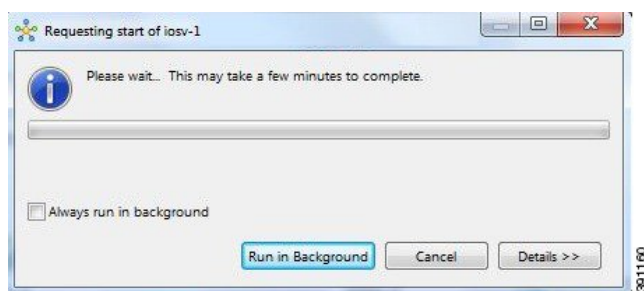
Step 2 Click **Start this node**.

Figure 131: Start This Node Option



The **Requesting start** dialog box appears.

Figure 132: Requesting Start Dialog Box



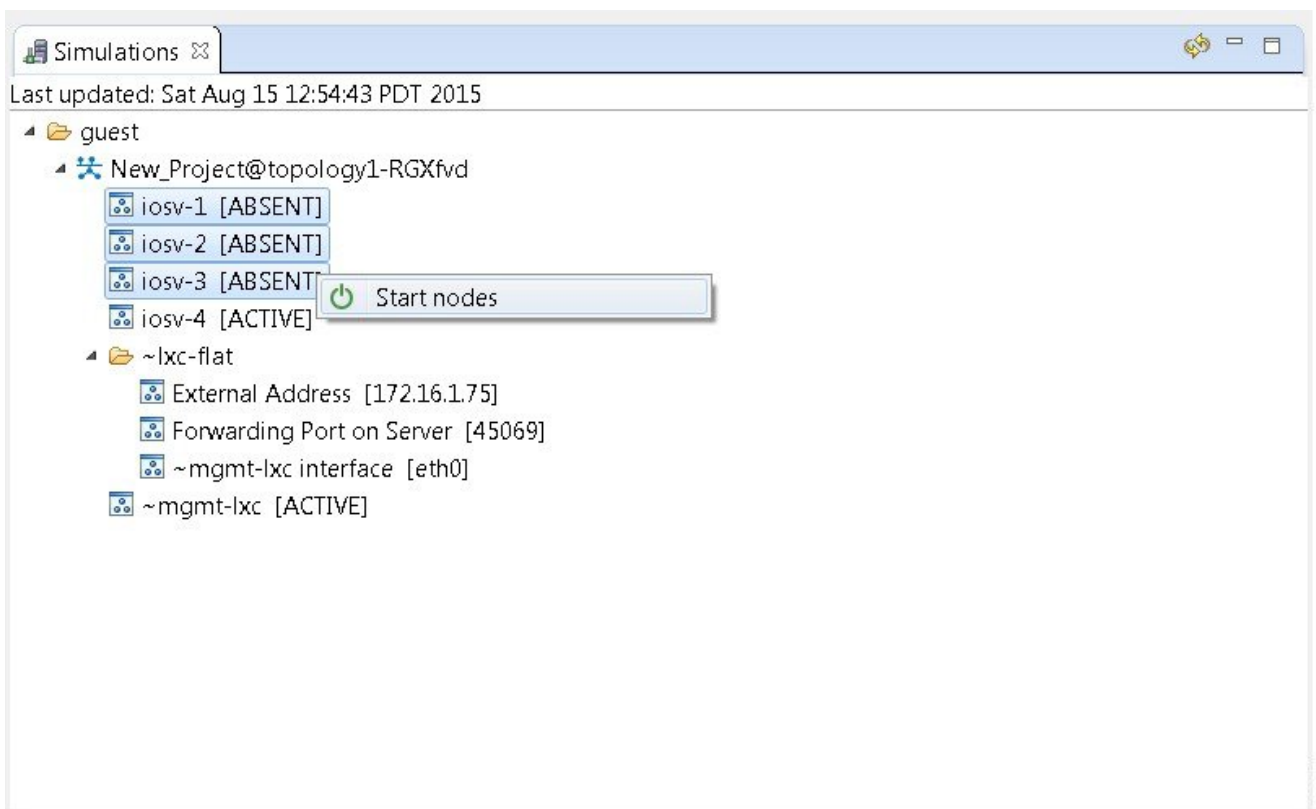
Note The status of the selected node is changed from **[ABSENT]** to **[ACTIVE]**, indicating that the node is up and running.

Start Multiple Nodes in a Running Simulation

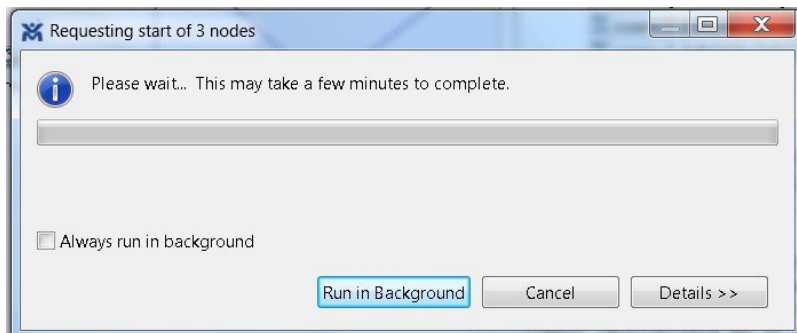
In cases where a phased simulation is running, you can later start those nodes not started with the initial simulation. To start multiple nodes, complete the following steps.

- Step 1** In the **Simulations** view, click the first node in the list to be started.
- Step 2** Hold down the **Shift** key and select the remaining nodes.
- Step 3** Right-click the selected nodes.
- Step 4** Click **Start nodes**.

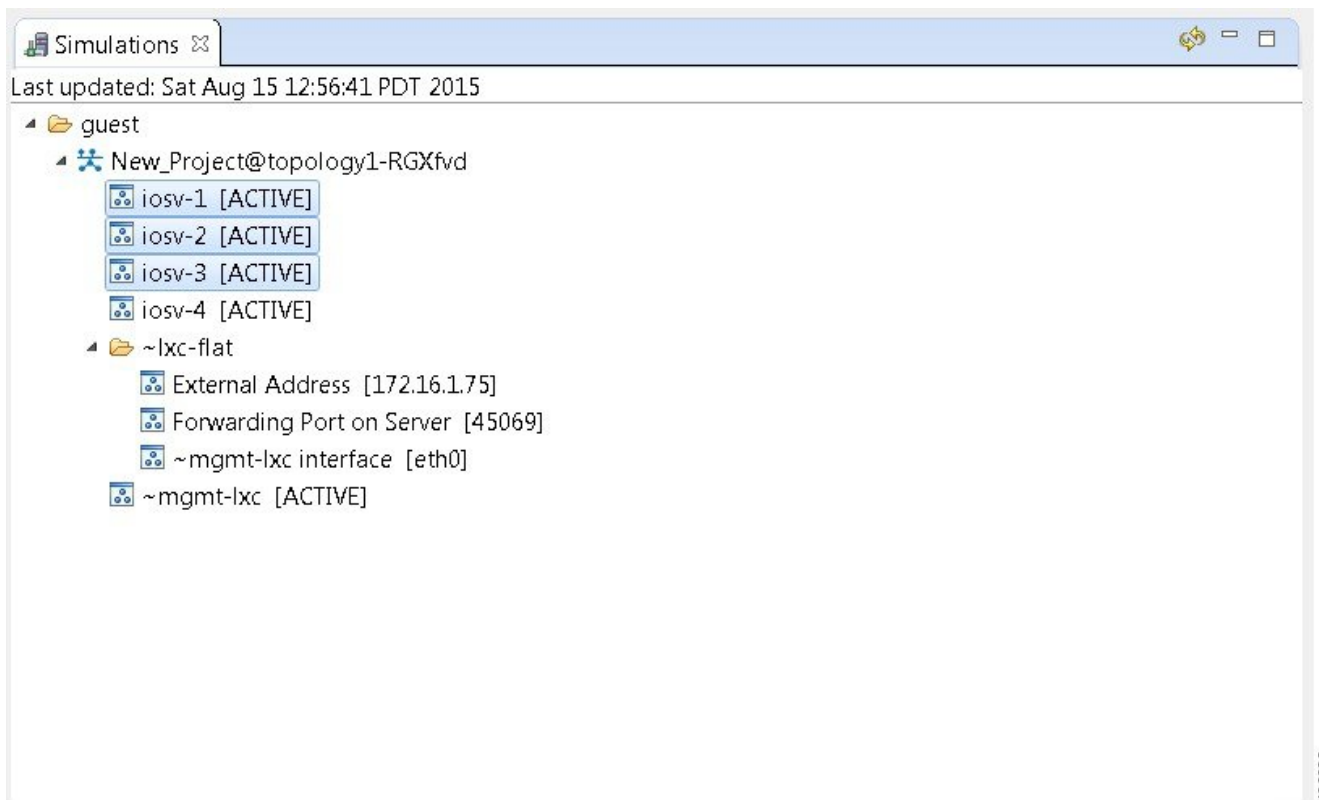
Figure 133: Start Nodes Option



The **Requesting start** dialog box appears.

Figure 134: Requesting Start Dialog Box

Note The status of the selected nodes changes from **[ABSENT]** to **[ACTIVE]**, indicating that the nodes are up and running.

Figure 135: Multiple Nodes Started

Stop a Simulation

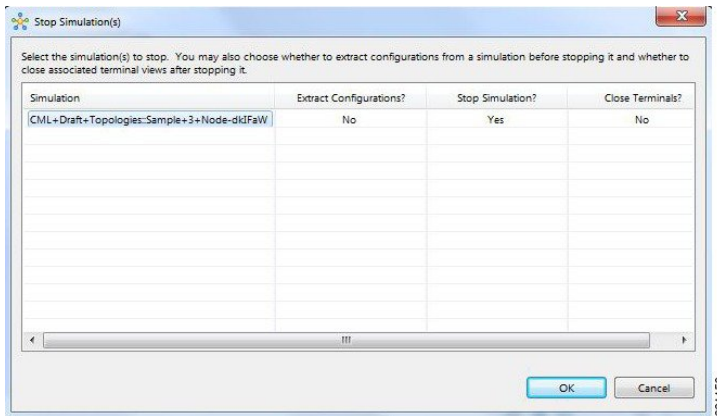
There are several ways to stop a simulation. In addition, you can stop multiple simulations at the same time. These are discussed in the following sections.

Stop a Simulation from the Toolbar

To stop a simulation from the toolbar, complete the following steps.

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

Figure 136: Stop Simulation(s) Dialog Box



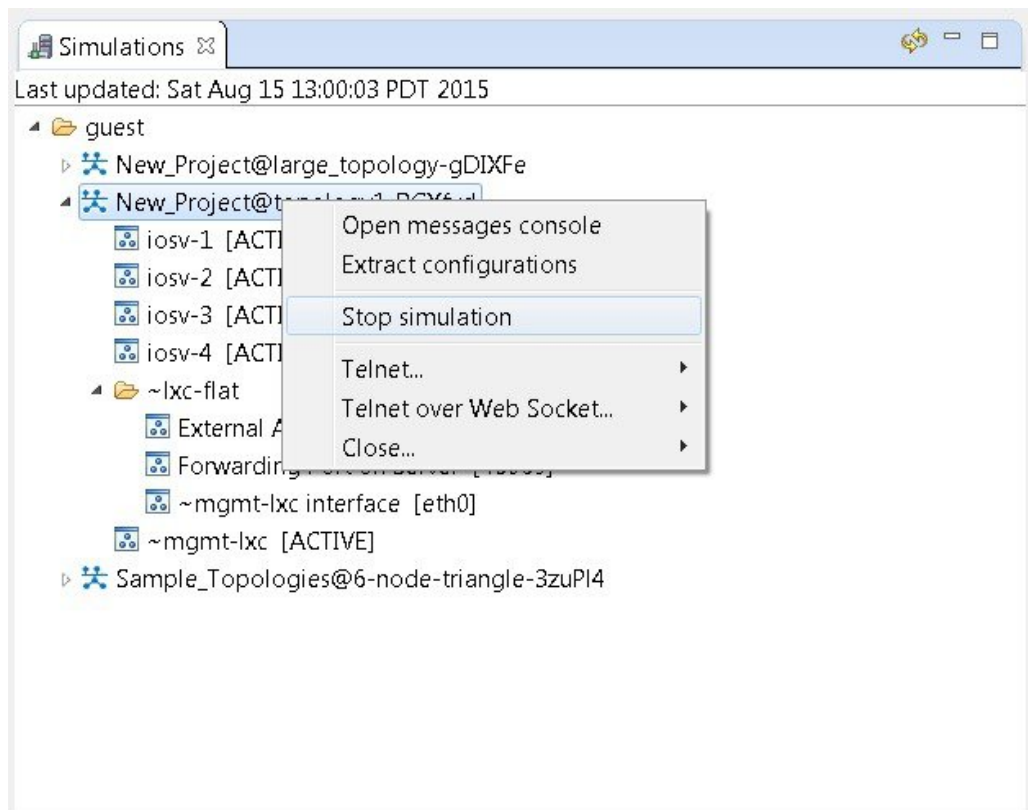
- Step 2** In the **Simulation** column, click once to highlight the simulation to stop.
- Step 3** (Optional) To save the configurations, click the adjacent setting in the **Extract Configurations?** column until the prompt changes to **Yes**.
Note Configurations for server nodes are not extracted.
- 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 stop the simulation, or click **Cancel** to leave the simulation running.
 On 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.

Stop a Simulation from the Simulations View

To stop a simulation, complete the following steps.

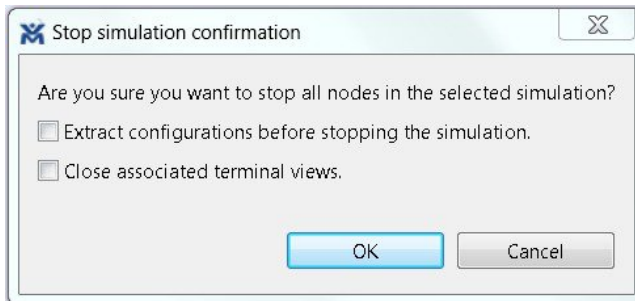
Step 1 In the **Simulations** view, right-click the simulation name and select **Stop Simulation**.

Figure 137: Stop a Simulation



The **Stop Simulation Confirmation** dialog box appears.

Figure 138: Stop Simulation Confirmation Dialog Box



- (Optional) Check the **Extract Configurations before Stopping the 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.

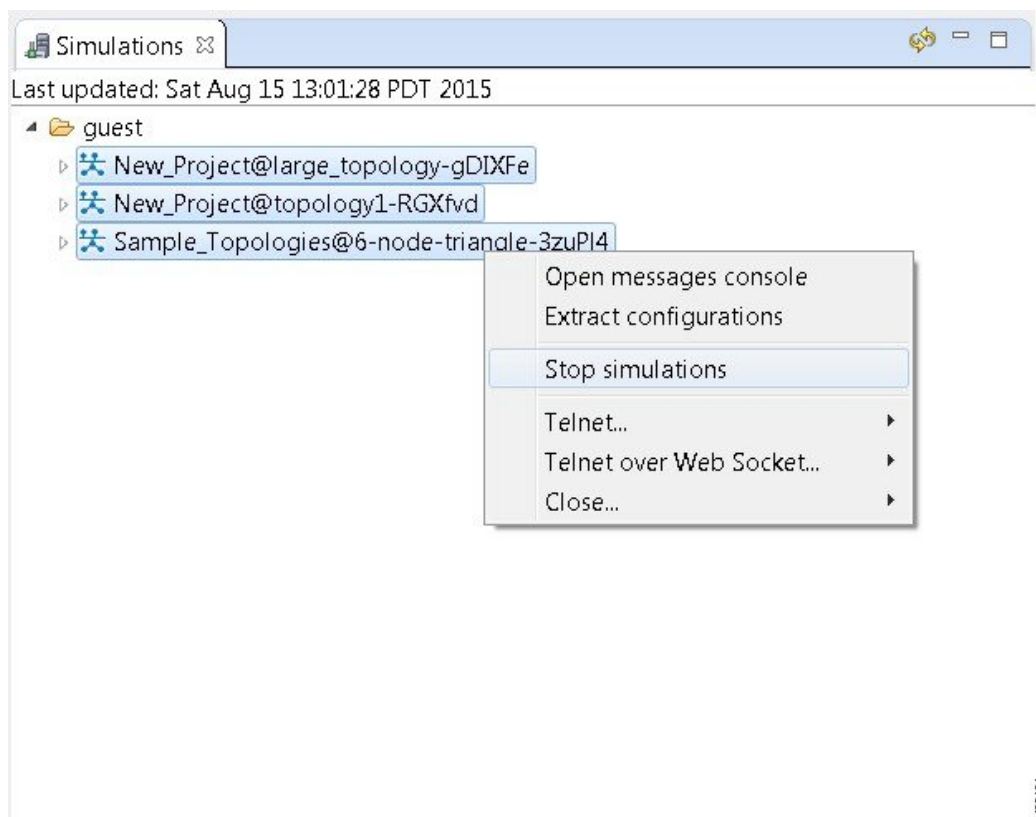
Note For instances where a user account expires, all running simulations for that user continue to run. Since the user account can no longer log in to stop them, they will remain active until the next system reboot or until the system administrator explicitly stops them.

Stop Multiple Simulations from the Simulations View

To stop multiple simulations, complete the following steps.

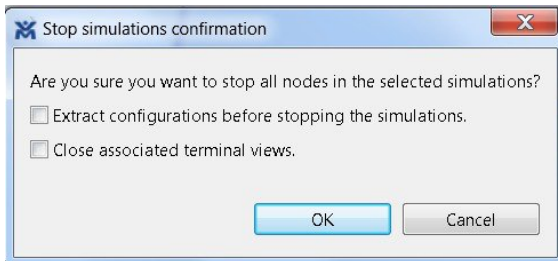
- Step 1** In the **Simulations** view, click the first simulation in the list to stop.
- Step 2** Hold down the **Shift** key and select the remaining simulations.
- Step 3** Right-click the selected simulations and select **Stop Simulations**.

Figure 139: Stop Multiple Simulations



The **Stop Simulations Confirmation** dialog box appears.

Figure 140: Stop Simulations Confirmation Dialog Box



- (Optional) Check the **Extract Configurations before Stopping the 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 4 Click **OK** to stop the simulations.

Once selected, all nodes in the simulations start shutting down. It may take a few minutes for the simulations to shut down completely and to disappear from the **Simulations** view.

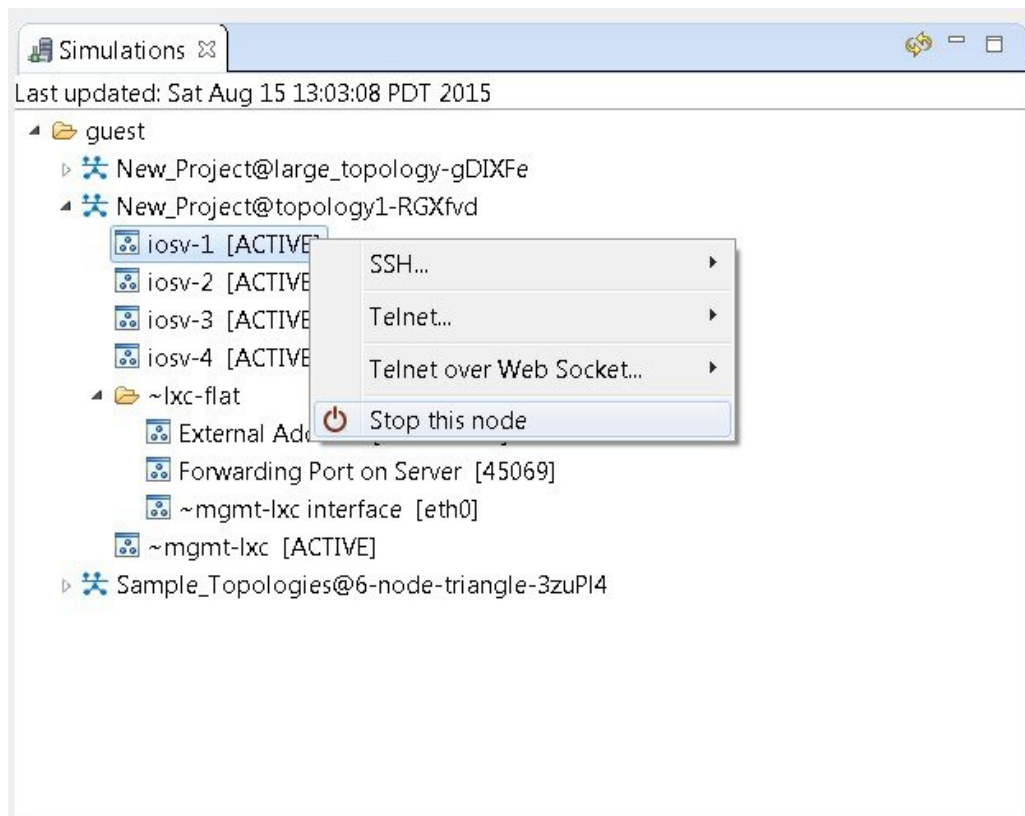
Note For instances where a user account expires, all running simulations for that user continue to run. Since the user account can no longer log in to stop them, they will remain active until the next system reboot or until the system administrator explicitly stops them.

Stop a Single Node

To stop a single node in a simulation, complete the following steps.

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

Figure 141: Stop a Single Node



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

- Step 2** Click **OK** to stop the node. 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.

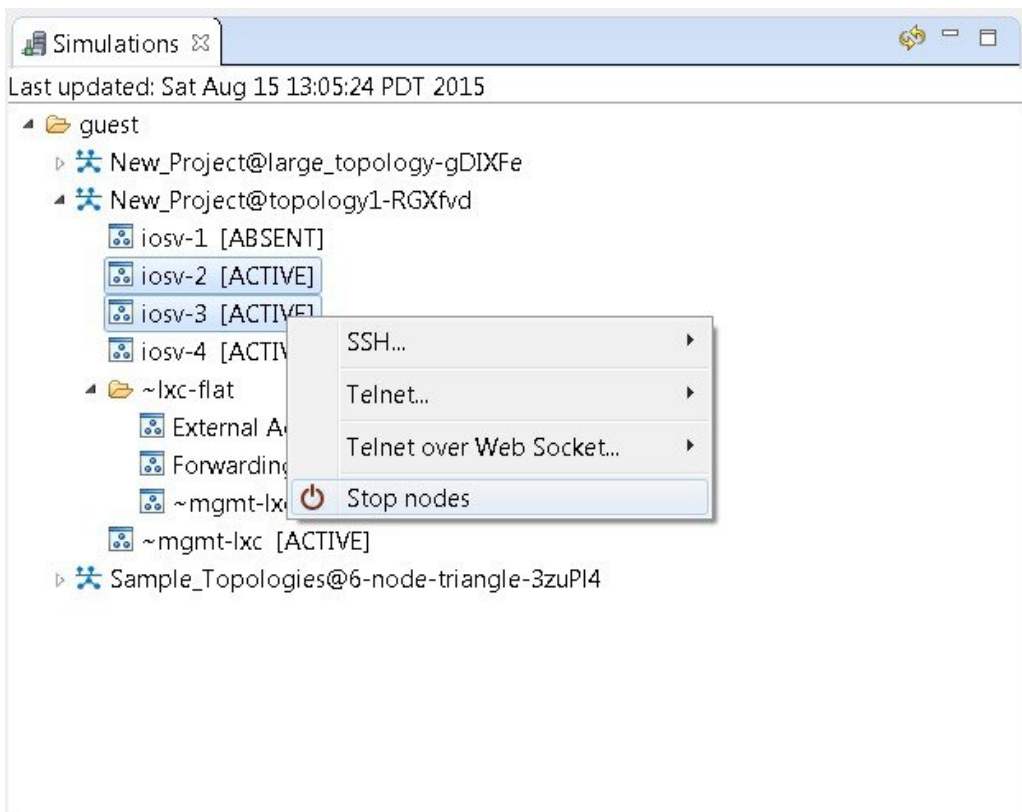
When a node is stopped, its status changes to **[ABSENT]**.

Stop Multiple Nodes

To stop multiple nodes in a running simulation, complete the following steps.

- Step 1** In the **Simulations** view, click the first node in the list to stop.
- Step 2** Hold down the **Shift** key and select the remaining nodes.
- Step 3** Right-click the selected nodes.
- Step 4** Click **Stop Nodes**.

Figure 142: Stop Multiple Nodes



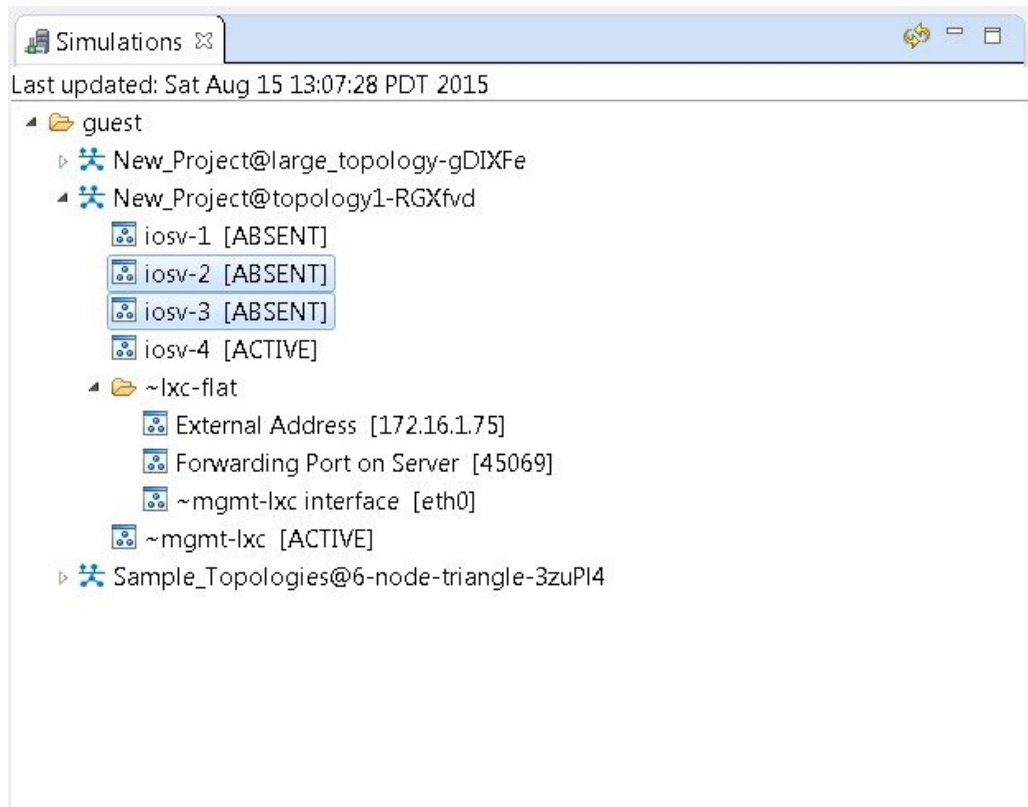
The **Are you sure?** dialog box appears.

Step 5 Click **OK** to stop the nodes. Alternatively, click **Cancel** to abandon the operation and return to the simulation.

Note When you click **OK**, the nodes stop without saving any changes to the configuration.

When the nodes are stopped, their status changes to [ABSENT].

Figure 143: Multiple Nodes Stopped



Modify a Node Configuration in the Simulation

You can modify node configurations in a running simulation. To do this, the type of connection available, either SSH or Telnet depends on the option set for the topology property **Management Network**.

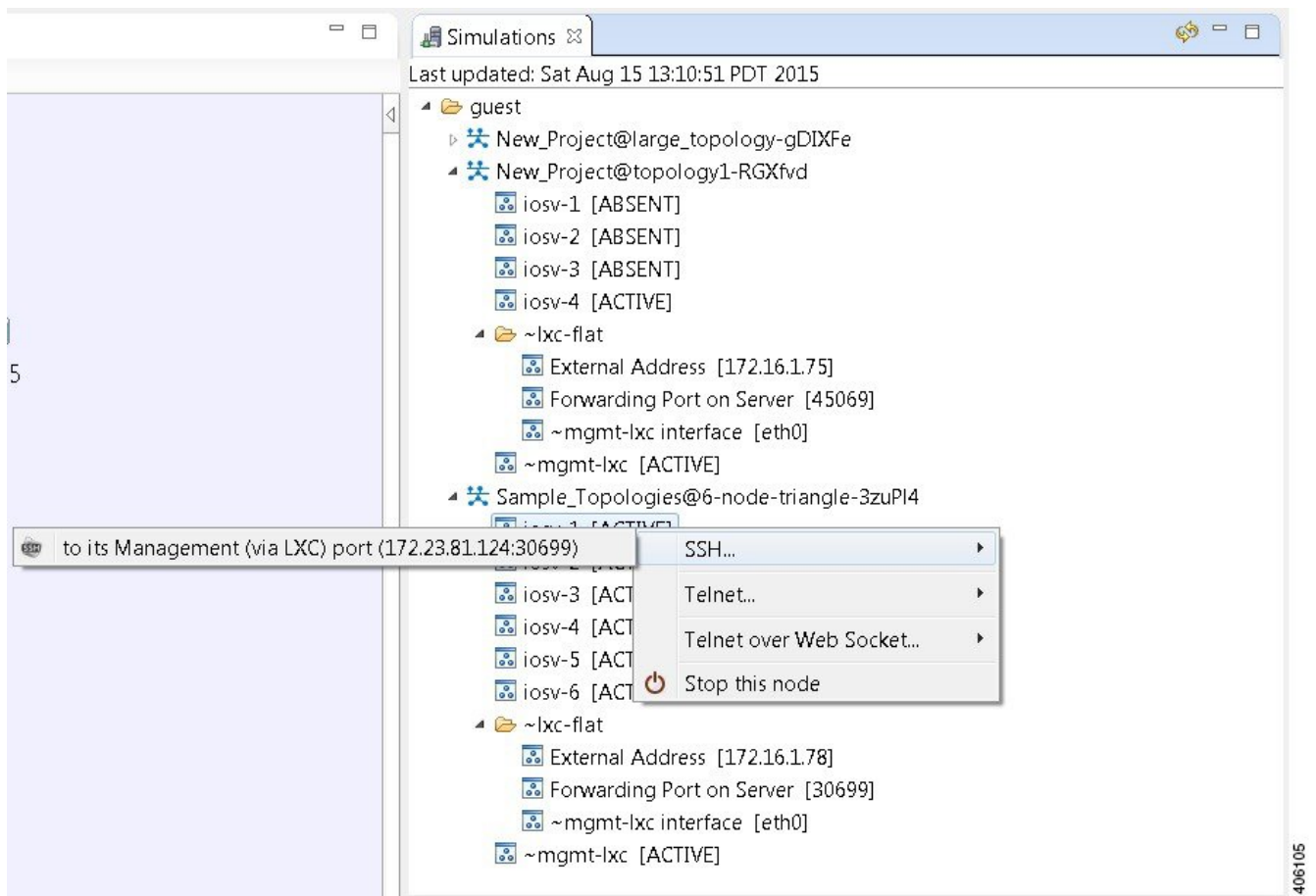
- When **Management Network** is set to **Private simulation network**, SSH and Telnet are available.
- When **Management Network** is set to **Shared flat network**, **Private project network**, or **not specified**, only Telnet is available.

Modify a Node Configuration in the Simulation via SSH

To modify a node configuration in a running simulation via SSH, complete the following steps.

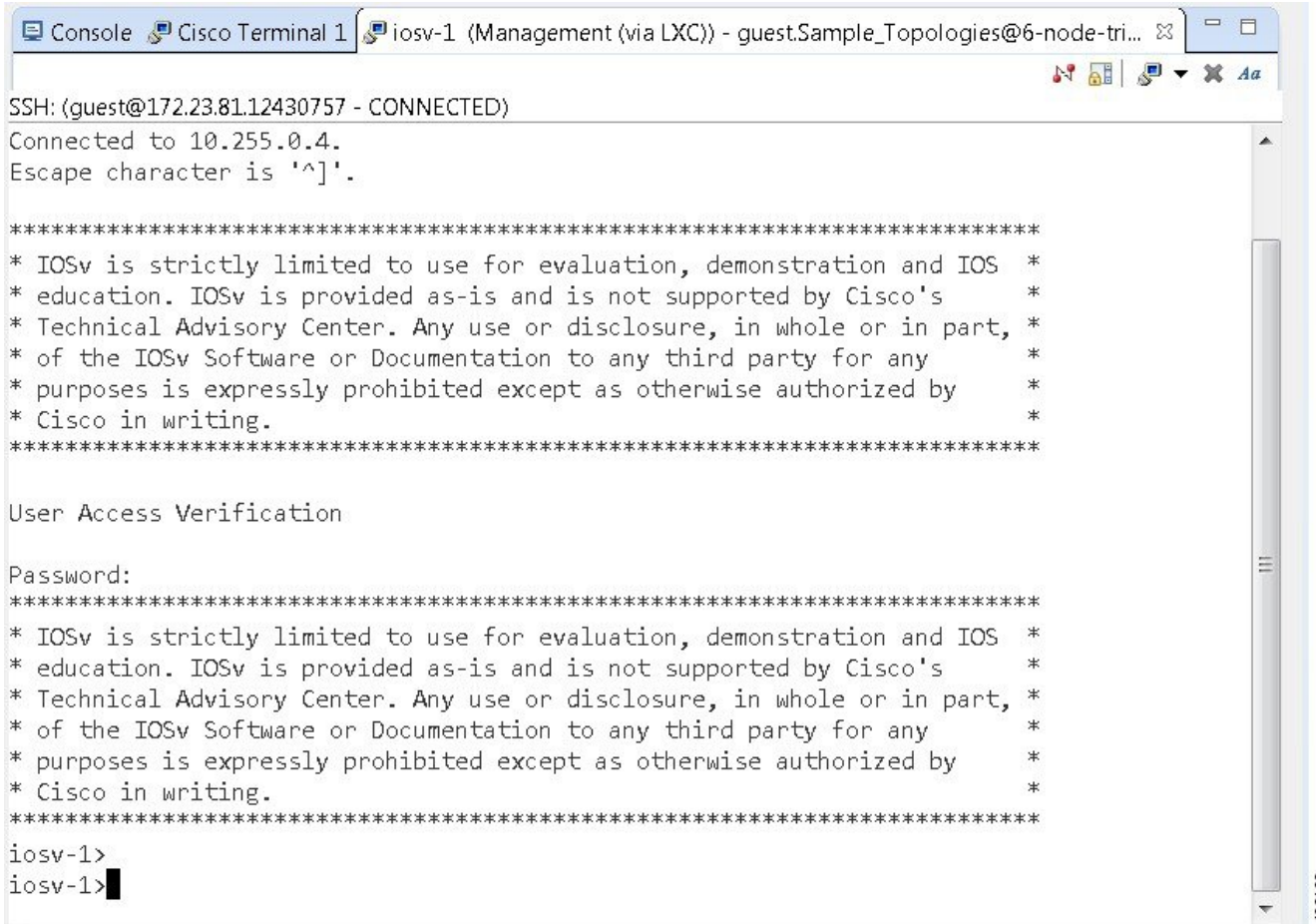
Step 1 Right-click the node in the **Simulations** view and choose **SSH > to its Management (via LXC) port**.

Figure 144: Connecting to a Node Console



A new **Terminal** view opens.

Figure 145: Terminal View



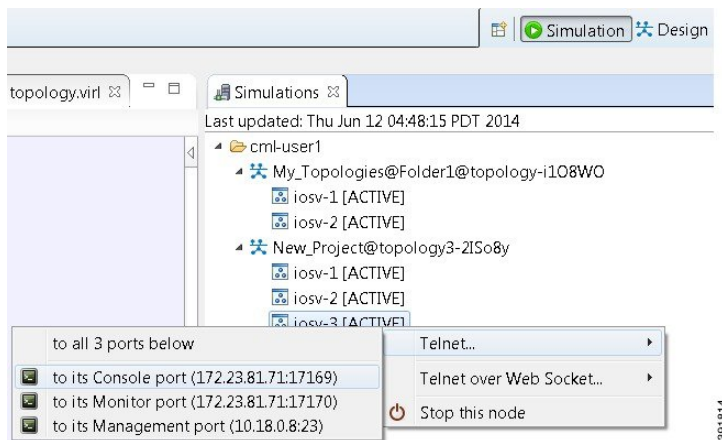
- Step 2** If no banner or router prompt is visible, press **Enter**.
You are now working with the operating system running on the node, for example, Cisco IOSv virtual software.
 - Step 3** Use the operating system commands to view or modify the node configuration.
- Note** Changes you make to the configuration do not appear in the canvas of the **Topology Editor**.

Modify a Node Configuration in the Simulation via Telnet

To modify a node configuration in a running simulation via Telnet, complete the following steps.

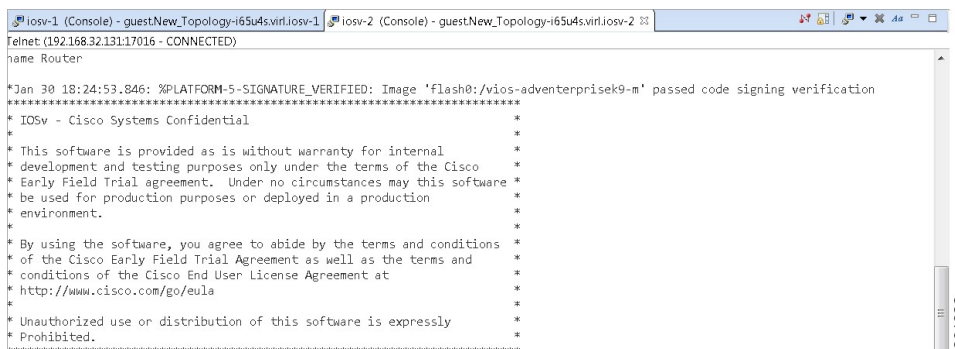
Step 1 Right-click the node in the **Simulations** view and choose **Telnet > to its Console port**.

Figure 146: Connecting to a Node Console



A new **Terminal** view opens.

Figure 147: 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 IOSv virtual software.

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

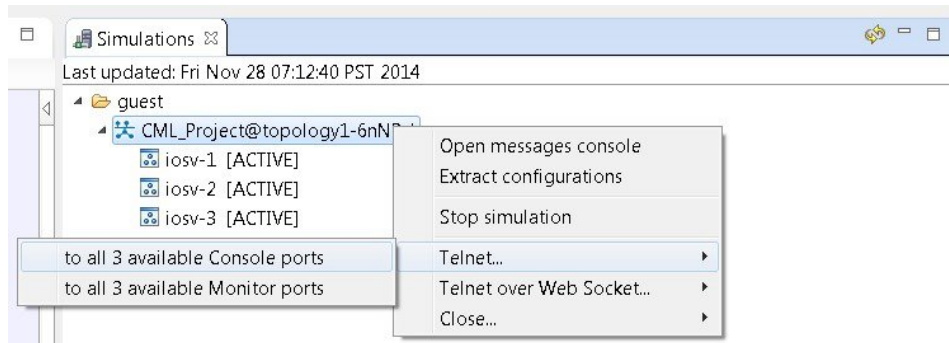
Note Changes you make to the configuration do not appear in the canvas of the **Topology Editor**.

Modify Multiple Node Configurations in the Simulation

To modify multiple node configurations in a running simulation, complete the following steps.

Step 1 Right-click the topology in the **Simulations** view and choose **Telnet > to all <number> available Console ports**.

Figure 148: Connect to Multiple Node Consoles



A new **Terminal** view opens for each of the consoles.

Figure 149: Terminal Views

```

Telnet: (172.23.81.124:17147 - CONNECTED)

*****
* IOSv is strictly limited to use for evaluation, demonstration and IOS *
* education. IOSv is provided as-is and is not supported by Cisco's *
* Technical Advisory Center. Any use or disclosure, in whole or in part, *
* of the IOSv Software or Documentation to any third party for any *
* purposes is expressly prohibited except as otherwise authorized by *
* Cisco in writing.
*****

iosv-1>
iosv-1>
iosv-1>en
Password:
iosv-1#

```

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 IOSv virtual 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

To extract and save modified configurations, complete the following steps.

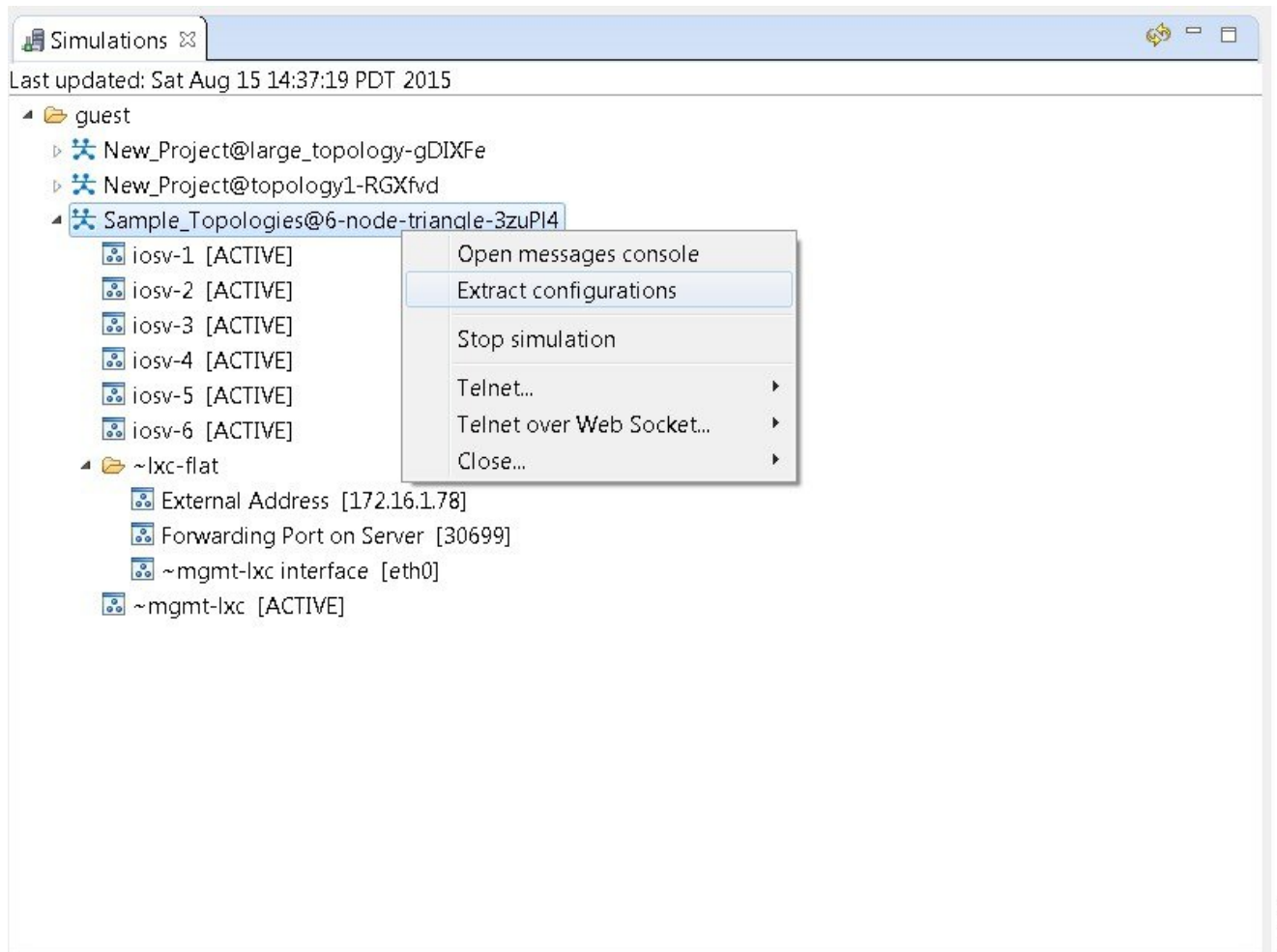
Before You Begin

- 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 150: Simulations View Context Menu

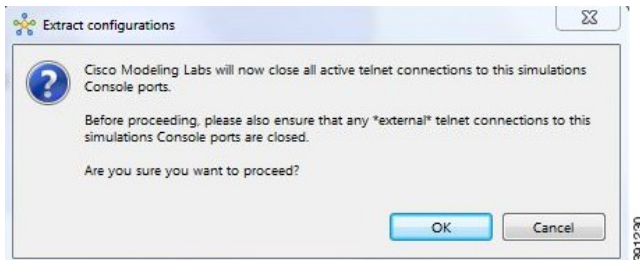


A confirmation dialog box appears.

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 151: Extract Configurations Dialog Box



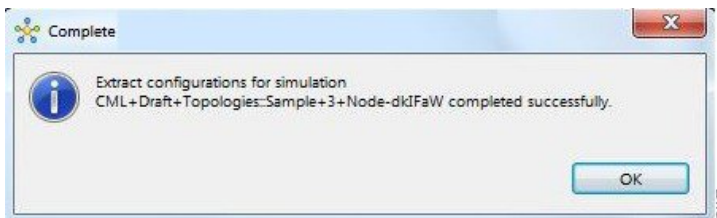
Step 3

Click **OK**.

The Extracting Configurations dialog box appears indicating that the extraction is in process.

When the extraction is complete, a message is displayed.

Figure 152: Extraction Complete Message



Note The configuration information is extracted and saved in the filename.virl 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 4

Click **OK**.

Partial Configuration Extraction

During a configuration extraction, if the process encounters issues or fails for a particular node, the problem node is identified and reported.

The extraction process then continues for all other nodes in the simulation and returns collected configurations to you.

Linux Server Snapshot Support

When a Linux server is present in a running simulation, you can use the **User Workspace Management** interface to take a snapshot of the disk content of the server. This newly created user-specific disk image can be used in other simulated sessions.

To take a snapshot of the server's disk contents, complete the following steps.

Step 1

Log in to the **User Workspace Management** interface.

Note You must log in as a user other than the uwmadmin user, for example, guest.

Step 2

On the **Overview** page, under **Sessions**, select the applicable running simulation.

Figure 153: Running Simulation Listed

User Workspace Management Logged in as guest Log out

Sessions

Session	Status	Expires	Options
~JumpHost	STOP	never	▼
CML_Project@topology1-Xc7wLD	ACTIVE	never	▼

Usage statistics for user guest

4 / 100 instances

1552 / 512000 RAM (MB)

4 / 200 VCPUS

A list of active VMs is displayed.

Figure 154: Active VMs

Session CML_Project@topology1-Xc7wLD details

Nodes

Node	Subtype	Status	Options
iosv-1	IOSv	ACTIVE	▼
iosv-2	IOSv	ACTIVE	▼
server-1	server	ACTIVE	▼
~mgmt-kxc	mgmt-kxc	ACTIVE	

Interfaces

Node	Interface name	Network subtype	Network name	IP Addresses
iosv-1	GigabitEthernet0/0	SESSION MGMT	mgmt	10.255.0.3 / 16
iosv-2	GigabitEthernet0/0	SESSION MGMT	mgmt	10.255.0.1 / 16
server-1	eth0	SESSION MGMT	mgmt	10.255.0.4 / 16
~mgmt-kxc	eth0	LXC FLAT	flat	172.16.1.147 / 24; public port: 30799
~mgmt-kxc	eth1	SESSION MGMT	mgmt	10.255.0.2 / 16

Step 3 Select the applicable Linux server, and from the **Options** drop-down menu, click **Create snapshot**.

Figure 155: Create Snapshot Option

Session CML_Project@topology1-Xc7wLD details

Sessions

Project sessions

Images

Flavors

Subtypes

Connectivity

Settings and password

Nodes

Node	Subtype	Status	Options
iosv-1	IOSv	ACTIVE	▼
iosv-2	IOSv	ACTIVE	▼
server-1	server	ACTIVE	▼
~mgmt-lxc	mgmt-lxc	ACTIVE	<ul style="list-style-type: none"> ▣ VNC console ▸ Serial port 📄 Create snapshot

Interfaces

Node	Interface name	Network subtype	Network name	IP Addresses
iosv-1	GigabitEthernet0/0	SESSION MGMT	mgmt	10.255.0.3 / 16
iosv-2	GigabitEthernet0/0	SESSION MGMT	mgmt	10.255.0.1 / 16
server-1	eth0	SESSION MGMT	mgmt	10.255.0.4 / 16
~mgmt-lxc	eth0	LXC FLAT	flat	172.16.1.147 / 24; public port: 30799
~mgmt-lxc	eth1	SESSION MGMT	mgmt	10.255.0.2 / 16

Project details for the newly created snapshot are displayed.

Figure 156: Newly Created Disk Image

User Workspace Management Logged in as guest Log out

Saving snapshot of node "server-1" as image "guest-server-server-1"...

Project image snapshot guest-server-server-1 details

Name
guest-server-server-1

OpenStack ID
e5da2898-4c68-4d3c-8a8b-0a649f905a8a

Project
d14434d936614631a0055da7deae8cca

Updated
2014-12-15 13:34:07

Status
queued

Size
0.00 B (0)

Minimum Disk Size
undefined

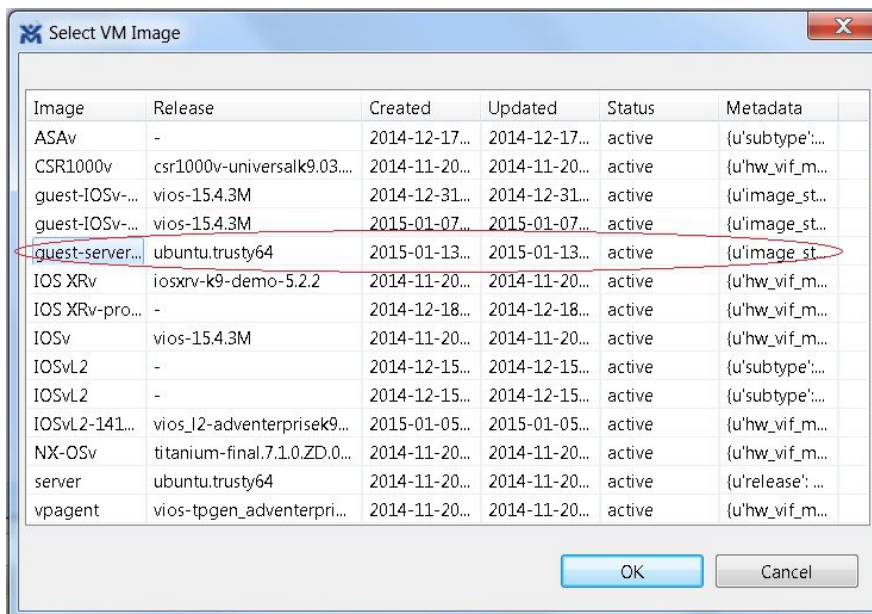
Converted qcow2 image checksum
None (every image is forced through conversion, even qcow2)

Reuse the Image Snapshot

To reuse the image snapshot, complete the following steps.

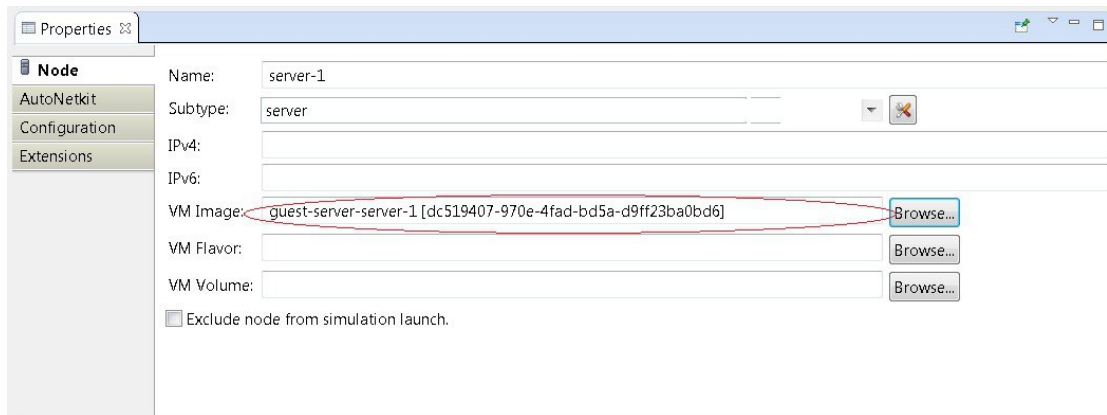
- Step 1** Create a new topology or open an existing topology.
- Step 2** On the canvas, add a node to the topology.
- Step 3** Select the node on the canvas.
The sample topology opens in the **Topology Editor** canvas.
- Step 4** In the **Properties > Node** view, click **Browse** beside the **VM Image** field.
The **Select VM Image** dialog box appears.
- Step 5** Select the applicable image snapshot and click **OK**.

Figure 157: Select the Image Snapshot to Use



Details for the image snapshot are visible in the **VM Image** field under **Properties > Node** view.

Figure 158: Image Snapshot Selected





External Connectivity in Cisco Modeling Labs

- [External Connectivity to a Node](#), page 179
- [Set Up a FLAT Network for Out of Band \(OOB\) Management Access](#), page 180
- [Set Up a FLAT Network for Inband Access](#), page 186
- [Set Up a SNAT Network for Inband Access](#), page 188

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 by communicating with the Cisco Modeling Labs server at its IP address that is bound to the relevant management port.

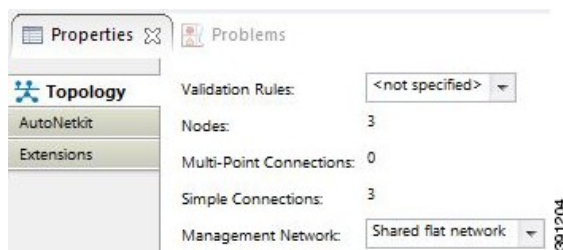
Set Up a FLAT Network for Out of Band (OOB) Management Access

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 [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.

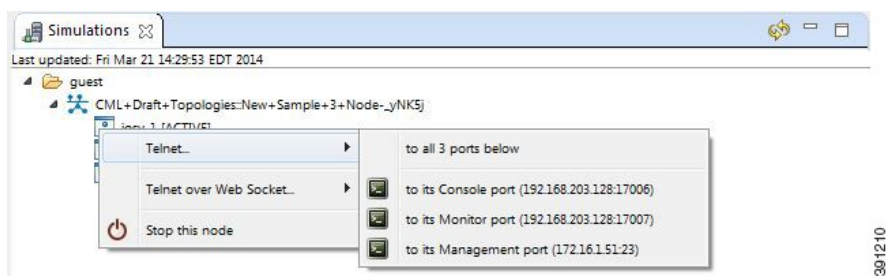
Step 6 From the toolbar, click the **Build Initial Configurations** tool.

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

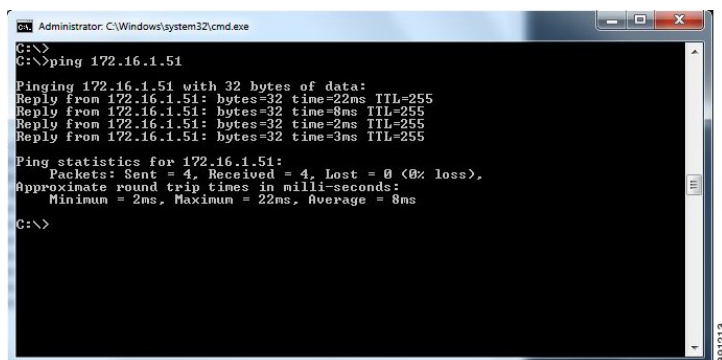
Step 7 After the build phase is completed, initiate the simulation by clicking the **Launch Simulation** tool in the toolbar, and then change to the **Simulation** perspective.

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 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.

```

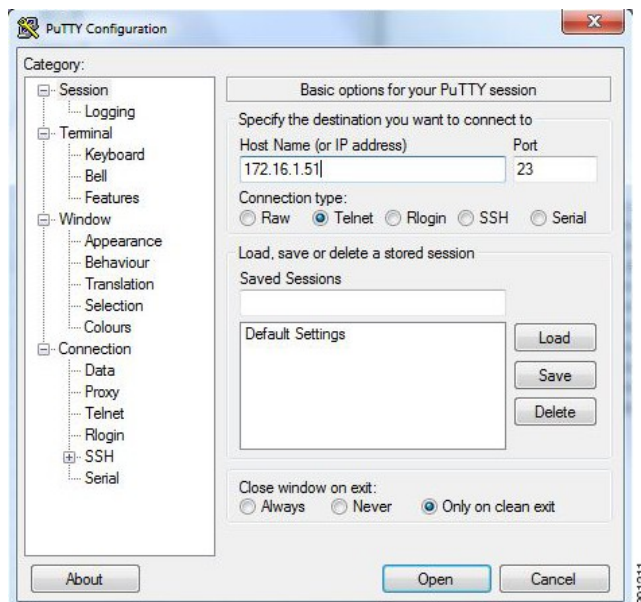
iosv-1 (Console) - guest.CML+Draft+Topologies::New+... Console Node Editor Progress
Telnet (192.168.203.128:17006 - CONNECTED)
* be used for production purposes or deployed in a production
* environment.
*
* By using the software, you agree to abide by the terms and conditions
* of the Cisco Early Field Trial Agreement as well as the terms and
* conditions of the Cisco End User License Agreement at
* http://www.cisco.com/go/eula
*
* Unauthorized use or distribution of this software is expressly
* Prohibited.
*****
iosv-1>
iosv-1>show ip interface brief
Interface                IP-Address      OK? Method Status  Protocol
GigabitEthernet0/0      172.16.1.51    YES NVRAM  up      up
GigabitEthernet0/1      10.0.0.1       YES NVRAM  up      up
GigabitEthernet0/2      10.0.0.5       YES NVRAM  up      up
GigabitEthernet0/3      unassigned     YES unset  up      up
Loopback0                192.168.0.1    YES NVRAM  up      up
iosv-1>
  
```

301214

Step 10

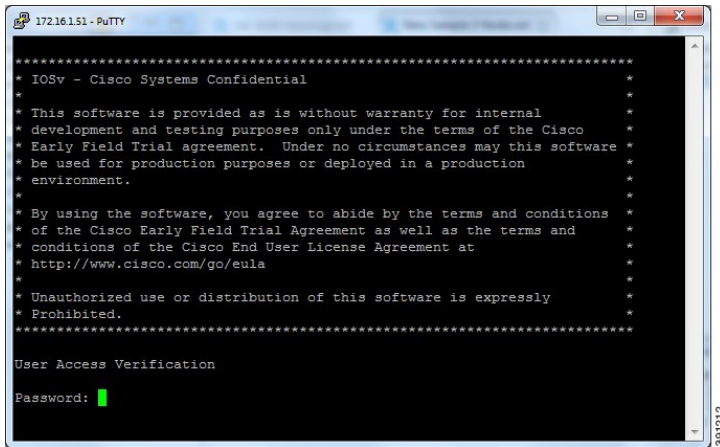
Launch and configure a terminal emulator for a Telnet connection to the node management port. In this example, the PuTTY application is used.

- 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.



301211

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

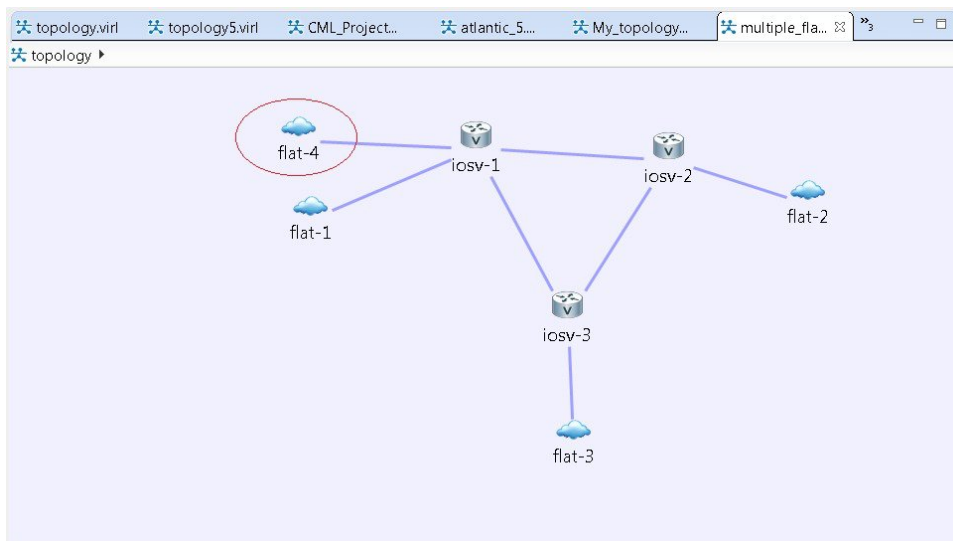


Add an Additional FLAT Network to a Simulation

The following procedure describes how to configure a second FLAT network to your simulation. A FLAT network uses Layer 2 connectivity, in which the IP address information about your virtual network can be viewed externally.

- Step 1** With your topology open on the canvas, click the **L2 External (FLAT)** icon to add another FLAT network to your topology, as shown.

Figure 159: Add an Additional FLAT Network



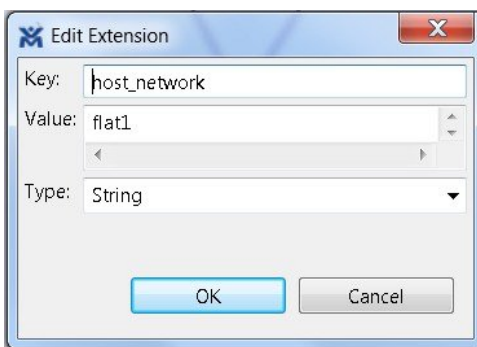
Step 2 Click the FLAT network just added, and in the **Extensions** tab in the **Properties** view, click the **Add new extension** icon.

The **Edit extension** dialog box is displayed.

Step 3 Add the following details for the new extension and click **OK**.

- **Key:** host_network
- **Value:** flat1
- **Type:** String

Figure 160: Edit Extension Dialog Box



Note You must add this new extension for the second FLAT network, as a second interface is not permitted in the same subnet. This FLAT network is in the **flat1** subnet.

Step 4 From the toolbar, click the **Build Initial Configurations** tool.

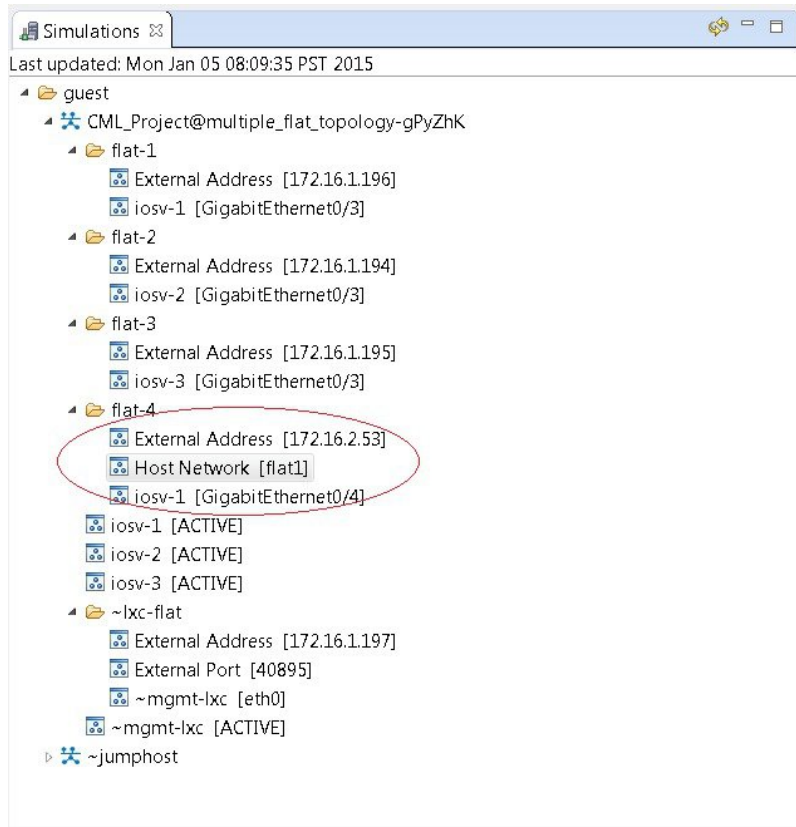
Note For the purpose of this task, you may retain the default settings.

During the simulation phase, the system automatically assigns an IP address to the node interface that is connected to the FLAT cloud.

Step 5 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.

The IP addresses assigned to each of the interfaces in the FLAT network and connected to each node, are displayed in the **Simulations** view. These IP addresses are automatically applied to the nodes when they become active.

Figure 161: Flat1 Subnet



Step 6 Regarding the node that has two FLAT network connections, when its state has changed to **[ACTIVE]**, log in to said node.

Step 7 Enter the command `show ip interface bri` to view all interfaces available to the node.

Figure 162: Viewing the FLAT Networks

```

iosv-1 (Management (via LXC)) - guest.CML_Project@multiple_fl... Console
SSH: (guest@172.23.81.12440953 - CONNECTED)

Password:
*****
* IOSv is strictly limited to use for evaluation, demonstration and
* education. IOSv is provided as-is and is not supported by Cisco's
* Technical Advisory Center. Any use or disclosure, in whole or in p
* of the IOSv Software or Documentation to any third party for any
* purposes is expressly prohibited except as otherwise authorized by
* Cisco in writing.
*****
iosv-1>en
Password:
iosv-1#show ip interface bri
Interface                IP-Address      OK? Method Status
-----
GigabitEthernet0/0      10.255.0.3      YES NVRAM  up
GigabitEthernet0/1      10.0.0.9        YES NVRAM  up
GigabitEthernet0/2      10.0.0.13       YES NVRAM  up
GigabitEthernet0/3      172.16.1.196   YES NVRAM  up
GigabitEthernet0/4      172.16.2.53    YES NVRAM  up
Loopback0                192.168.0.1    YES NVRAM  up

iosv-1#

```

You can see that the node has a connection into each of the external FLAT connectivity networks, 172.16.1.196 and 172.16.2.53.

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 [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 **Build Initial 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.

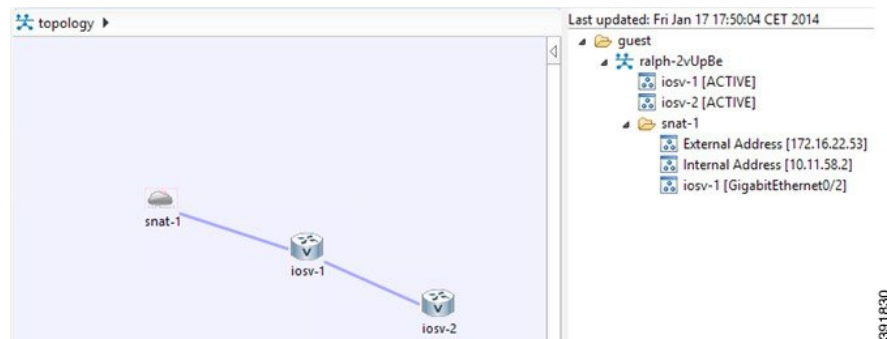
If these items are correct, ping to each of the key devices in the path to determine where the failure occurs and notify your system administrator of the failure source if it is outside your Cisco Modeling Labs environment.

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 163: 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 [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 **Build Initial 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.
