

WANDL

Training Guide for IP/MPLSView

Release

6.4.0



30 August 2016

Juniper Networks, Inc.
1194 North Mathilda Avenue
Sunnyvale, California 94089
USA
408-745-2000
www.juniper.net

Juniper Networks, Junos, Steel-Belted Radius, NetScreen, and ScreenOS are registered trademarks of Juniper Networks, Inc. in the United States and other countries. The Juniper Networks Logo, the Junos logo, and JunosE are trademarks of Juniper Networks, Inc. All other trademarks, service marks, registered trademarks, or registered service marks are the property of their respective owners.

Juniper Networks assumes no responsibility for any inaccuracies in this document. Juniper Networks reserves the right to change, modify, transfer, or otherwise revise this publication without notice.

WANDL Training Guide for NPAT and IP/MPLSView
Copyright © 2016, Juniper Networks, Inc.
All rights reserved.

The information in this document is current as of the date on the title page.

YEAR 2000 NOTICE

Juniper Networks hardware and software products are Year 2000 compliant. Junos OS has no known time-related limitations through the year 2038. However, the NTP application is known to have some difficulty in the year 2036.

END USER LICENSE AGREEMENT

The Juniper Networks product that is the subject of this technical documentation consists of (or is intended for use with) Juniper Networks software. Use of such software is subject to the terms and conditions of the End User License Agreement ("EULA") posted at <http://www.juniper.net/support/eula.html>. By downloading, installing or using such software, you agree to the terms and conditions of that EULA.

Table of Contents

About the Documentation	xi
Documentation and Release Notes	xi
Documentation Feedback	xi
Requesting Technical Support	xi
Self-Help Online Tools and Resources	xii
Opening a Case with JTAC	xii
Chapter 1 Introduction	1
Following Along With the Examples in This Manual	1
Key Software Features	2
Add-On Features	3
Supported Hardware	3
Alcatel	3
Cisco Systems	3
ECI Telecom LTD	3
Force10	3
Foundry	3
Huawei	4
IBM	4
Juniper Networks	4
Lucent Technologies (Ascend Communications)	4
Marconi	4
net.com	4
Nortel	4
Riverstone	4
Sycamore	4
Tellabs	4
RAD	4
Converting Your Files to IP/MPLSView File Format	4
Extraction Tools for Topology and Provisioned Data	5
Extraction Tools for Traffic Data	5
Chapter 2 Documentation Conventions	7
Document Conventions	7
Keyboard, Window, and Mouse Terminology and Functionality	7
The Keyboard	8
The Mouse	8
Informational Notes, Cautions, and Warnings	8
Changing the Size of a Window	9
Moving a Window	9
For Further Information	9
Chapter 3 Understanding the Specification File	11
Prerequisites	11
Related Documentation	11

Definition of a Specification File	11
Specification File Format	11
runcode	12
datadir	12
File Types.	12
File Names	12
Creating Input Files	12
Example Set of Network Files	13
Adding Comments.	14
Additional Notes	14
datadir - Specifying the Location of Input Files	14
Making a Copy of a Specification File	15
Chapter 4 Opening and Closing a Network Project.	17
When to use	17
Prerequisites	17
Related Documentation.	17
Recommended Instructions.	17
Detailed Procedures	17
Open Network Project File	17
Close Network Project File	19
Accessing Recently Opened Project Files	19
Chapter 5 Creating and Editing a Network Model	21
Definition of a Specification File	21
When to use	21
Recommended Instructions.	21
Detailed Procedures	22
Creating a Directory for Your Network	22
Create a Specification File	22
Editing a Specification File	24
Creating or Editing a Network File via the Text Editor.	24
Chapter 6 Loading a Network File.	27
When to use	27
Recommended Instructions.	27
Detailed Procedures	28
Chapter 7 Using the Topology Window	29
When to use	29
Prerequisites	29
Related Documentation.	29
Recommended Instructions.	29
Select and Move Elements	30
Graphical vs. Geographical Coordinates	30
Element Selection	30
Using the Drag/Rotate Tool	31
Network Information Window	32
Finding Network Elements on the Map	33
Using the Legends	34
Utilization legend	34

Customizing Nodes and Links in the Legends	36
Network Element Labels	38
Node Labels	38
Link Labels	38
Bottom Bar Labels	39
Displaying Geographical Maps	39
Map Preferences	40
Inverting Colors	41
Topology Layout	42
Distribute Selected Points	42
Circle Nodes	43
Straighten Nodes	43
Reset by Lat/Lon	43
Cut, Copy, and Paste	43
Zooming In and Out and Using the Navigator Window	44
Grouping Functions	44
Viewing Your Network Model By Subview	46
Map Views	46
Copy Map Layout	47
Export... and Print...	47
Print...	47
Export...	48
Export to Visio	48
Export Map to Visio	48
Setting Up the Stencils (for Visio 2002 and earlier only)	49
Opening the Visio Drawing	50
Page Setup	50
Setting Link Colors and Curvature	51
Handling Groups	51
Visio Tips	52
Scalable Vector Graphics	53
Chapter 8 Viewing and Saving Network Information	55
When to use	55
Prerequisites	55
Related Documentation	55
Recommended Instructions	55
Detailed Procedures	56
Node, Link, and Demand Searches	57
Advanced Search	59
Node, Link, and Demand Map Menus	59
Saving Network Info to a Report	61
Viewing Demands	62
Obtaining and Saving Network Information in the Console	63
Performing a Path Analysis	64
Viewing Aggregate Demand Information	66
Chapter 9 Adding Nodes, Links, and Demands	67
When to use	67
Prerequisites	67
Related Documentation	67

Recommended Instructions	67
Detailed Procedures	68
Adding a Node	68
Adding a Link	69
Adding Demands	71
Saving Network	73
Sample Files	73
spec.triangle3	73
dparam.triangle3	73
muxloc.triangle3	73
bblink.triangle3	74
demand.triangle3	74
graphcoord.triangle3	74
nodeparam.triangle3	74
Deriving npa nxx from CLLI codes using wanpricer*	74
Customizing the Location Database	75
Adding a Country	75
Creating a City Database for a Country	75
Chapter 10 Modifying Nodes, Links and Demands	77
When to use	77
Prerequisites	77
Related Documentation	77
Recommended Instructions	77
Detailed Procedures	77
Modifying Nodes	77
Modifying Links	79
Modifying Demands	80
Deleting Nodes, Links, and Demands	81
Chapter 11 Modifying Sites and Owners	83
When to use	83
Prerequisites	83
Related Documentation	83
Recommended Instructions	83
Detailed Procedures	84
Creating a Site	84
Creating Owners	86
Admin Weight/Metric Rules	88
Specifying Link Characteristics	88
Specifying a Formula Using Fixed and Variable Weight	88
Setting Multiple Formulas	88
Service Type	90
Chapter 12 User Parameters	93
When to use	93
Prerequisites	93
Related Documentation	93
Outline	93
User Parameters	94
User Parameters Window	94

	Modifying User Parameter Values	95
	Performing Text Filter on User Parameters	95
	Map Labels with User Parameters	96
	The Usrparam File Format	96
	Attribute Declarations	96
	Attribute/Value Pairs for Nodes and Links	96
	Attribute/Value Pairs for Tunnels	97
Chapter 13	Modifying Link Pricing	99
	When to use	99
	Prerequisites	99
	Related Documentation	99
	Recommended Instructions	99
	Detailed Procedures	100
	Custom Rates	100
Chapter 14	Design	103
	When to use	103
	Prerequisites	103
	Related Documentation	103
	Recommended Instructions	103
	Detailed Procedures	103
	The Resize Feature	103
	Create Simplified Group Model	106
	Simplified Group Model Walk-Through	106
Chapter 15	Simulation	109
	Backbone Simulation	109
	When to use	109
	Prerequisites	110
	Related Documentation	110
	Recommended Instructions	110
	Detailed Procedures	111
	Interactive Simulation	111
	Interactive Link Failure	111
	Interactive Node Failure	113
	Interactive SRLG/Facility Failure	113
	Stepping Through a Simulation	114
	View Changes	116
	Simulation Scenarios	117
	Exhaustive Single Element Failure Scripts	117
	Exhaustive Double and Triple Element Failures	118
	Simulation Reports	118
	Interpreting the Link Peak Utilization Report	119
	Generating One Report Per Failure	120
	Viewing Simulation Results Graphically	121
Chapter 16	Path Placement	123
	When to use	123
	Prerequisites	123
	Related Documentation	123

Recommended Instructions	124
Detailed Procedures	124
Viewing Path Placement Options	124
Specifying Routing Methods	124
Defining Placement Order	125
Path Selection Method	125
Understanding Routing Paths	125
Design from Scratch	125
Tuning	125
Restore Paths to Routing in Demand File	125
Network Grooming	125
Path Diversity Design	126
Chapter 17 Route Optimization	127
When to use	127
Related Documentation	127
Recommended Instructions	127
Detailed Procedures	127
Using the Route Optimization Wizard	127
Chapter 18 Network Grooming	129
When to Use	129
Prerequisites	129
Related Documentation	129
Recommended Instructions	129
Detailed Procedures	130
Selecting Demands for Net Groom	130
Setting Options for Net Groom	131
Optimizing Demands	132
Chapter 19 Report Manager	135
When to use	135
Prerequisites	135
Related Documentation	135
Recommended Instructions	135
Detailed Procedures	136
Report Options	136
Navigating the Report Manager	136
Customizing the Report View	137
Saving Reports	138
Chapter 20 Text Mode	139
When to use	139
Prerequisites	139
Recommended Instructions	139
Detailed Procedures	140
Running Bbdsgn	140
Navigation in the menu	140
Read Files	141
Save Files	142
Saving Keystrokes for Batch Mode	144

Tracing Input 144

Feeding Keystrokes 144

Creating Your Own Keystrokes Input 144

About the Documentation

Documentation and Release Notes

To obtain the most current version of all Juniper Networks® technical documentation, see the product documentation page on the Juniper Networks website at <http://www.juniper.net/techpubs/>.

If the information in the latest release notes differs from the information in the documentation, follow the product Release Notes. Juniper Networks Books publishes books by Juniper Networks engineers and subject matter experts. These books go beyond the technical documentation to explore the nuances of network architecture, deployment, and administration. The current list can be viewed at <http://www.juniper.net/books>.

Documentation Feedback

We encourage you to provide feedback, comments, and suggestions so that we can improve the documentation. You can provide feedback by using either of the following methods:

- Online feedback rating system—On any page at the Juniper Networks Technical Documentation site at <http://www.juniper.net/techpubs/index.html>, simply click the stars to rate the content, and use the pop-up form to provide us with information about your experience. Alternately, you can use the online feedback form at <https://www.juniper.net/cgi-bin/docbugreport/>.
- E-mail—Send your comments to techpubs-comments@juniper.net. Include the document or topic name, URL or page number, and software version (if applicable).

Requesting Technical Support

Technical product support is available through the Juniper Networks Technical Assistance Center (JTAC). If you are a customer with an active J-Care or JNASC support contract, or are covered under warranty, and need post-sales technical support, you can access our tools and resources online or open a case with JTAC.

- JTAC policies—For a complete understanding of our JTAC procedures and policies, review the JTAC User Guide located at <http://www.juniper.net/us/en/local/pdf/resource-guides/7100059-en.pdf>.
- Product warranties—For product warranty information, visit <http://www.juniper.net/support/warranty/>.
- JTAC hours of operation—The JTAC centers have resources available 24 hours a day, 7 days a week, 365 days a year.

Self-Help Online Tools and Resources

For quick and easy problem resolution, Juniper Networks has designed an online self-service portal called the Customer Support Center (CSC) that provides you with the following features:

- Find CSC offerings: <http://www.juniper.net/customers/support/>
- Search for known bugs: <http://www2.juniper.net/kb/>
- Find product documentation: <http://www.juniper.net/techpubs/>
- Find solutions and answer questions using our Knowledge Base: <http://kb.juniper.net/>
- Download the latest versions of software and review release notes: <http://www.juniper.net/customers/csc/software/>
- Search technical bulletins for relevant hardware and software notifications: <http://kb.juniper.net/InfoCenter/>
- Join and participate in the Juniper Networks Community Forum: <http://www.juniper.net/company/communities/>
- Open a case online in the CSC Case Management tool: <http://www.juniper.net/cm/>
- To verify service entitlement by product serial number, use our Serial Number Entitlement (SNE) Tool: <https://tools.juniper.net/SerialNumberEntitlementSearch/>

Opening a Case with JTAC

You can open a case with JTAC on the Web or by telephone.

- Use the Case Management tool in the CSC at <http://www.juniper.net/cm/>.
- Call 1-888-314-JTAC (1-888-314-5822 toll-free in the USA, Canada, and Mexico). For international or direct-dial options in countries without toll-free numbers, see <http://www.juniper.net/support/requesting-support.html>.

Chapter 1

Introduction

This training guide covers the features in common between NPAT (Network Planning and Analysis Tools) and IP/MPLSView, such as capacity planning, simulations, traffic studies and network modifications for “what-if” scenario analyses.

For those who do not need to use the program’s graphical display, there is a **text mode** version of the software (referred to as “bbdsgr”, or backbone design). This is described in Chapter 20, Text Mode.

For an explanation of file input formats and output formats, see the File Format Reference for IP/MPLSView. For learning how to use the graphical user interface, see the remainder of this guide. For detailed window by window information, see the IP/MPLSView Java-Based Graphical User Interface Reference.

Following Along With the Examples in This Manual

Many of the chapters in this training guide use a sample network to illustrate step by step procedures that you can follow. These networks are located in the /u/wandl/sample or \$WANDL_HOME/sample folder on your server, where \$WANDL_HOME is the directory in which the server was installed.

In the sample directory /u/wandl/sample/IP, navigate to one of the subdirectories and double-click on a “spec” file. This opens the network project.



Informational Note: To examine your license, view the npatpw file located on your server, in /u/wanddb/sys/npatpw or \$WANDL_HOME/db/sys/npatpw. If your license has expired (see the line “expire_date=”), please contact Juniper support. Otherwise, proceed to the next step.

Key Software Features

IP/MPLSView is an integrated software package that addresses backbone topological design and simulation, as well as maintenance issues using TDM multiplexers, Routers, Frame Relay switches and ATM switches. Several hardware devices are currently modeled.

Feature	Description
Capacity Planning and Topological Design	<ul style="list-style-type: none">• The impact of adding new demands, or traffic, can be analyzed in a network model before they are placed onto the real network. IP/MPLSView can determine whether there is sufficient available capacity in the network and recommend places where capacity should be added.• A network can be automatically designed based on end-to-end transmission bandwidth requirements, starting from an existing set of backbone trunks ("Incremental Design") or from scratch ("Green-field Design"). Using powerful heuristic algorithms, IP/MPLSView can evaluate whether trunks can be pruned from the network without compromising diversity constraints. Potential bottlenecks are also easily detected. Further analysis capabilities include load balancing and end-to-end delay and loss calculations.
Hardware-Specific Device Library	<ul style="list-style-type: none">• Hardware-specific device models have been developed by Juniper, as a result of close working relationships with the major device vendors. This ensures the accuracy and precision of the model, as well as the support of many device-specific features.• IP/MPLSView includes various data extraction tools to convert network data (for example, router configuration files) into a format readable by IP/MPLSView. See Converting Your Files to IP/MPLSView File Format on page 4.
Path Assignments	For an existing backbone network, the optimal layout or path assignment can be determined which satisfies all demand constraints. Demand constraints may include media preferences, diverse paths and hardware bandwidth overhead calculations. Available paths can be found even when the actual hardware fails to find a path. If the hardware switch supports user-specified paths, then paths found using the model can be downloaded to the real network.
Failure Simulation	Exhaustive and customizable failure scenarios can be simulated to analyze and evaluate a networks' resiliency and topology.
Tariffs	IP/MPLSView can help reduce private line leasing costs (monthly recurring and non-recurring components) by producing designs that satisfy user requirements and constraints while maintaining a minimal cost. You can specify your own pricing tables which will then be integrated into IP/MPLSView to price out lines. This is used extensively in the program's heuristic least-cost design algorithms. The savings reaped normally translate to a quick return on investment.
Carrier Co-location	This feature (also termed LSA) can be used to evaluate whether additional savings may be achieved by co-locating the hardware with an IXC's POPs. The additional savings are realized in eliminating one or both of the local loop components from the private line calculation.
Detailed Design and Analysis Reports	As with all modules, detailed reports are generated to help you to quickly assess and summarize network-related data. These reports include trunk pricing, trunk utilization and demand paths. Reports can be generated in text format, CSV (comma separated) format, or web-friendly HTML.
Detailed Topology Views	Detailed topology views allow you to view your network nodes and links. These can be placed by geographical location, rearranged by hand, or automatically rearranged in a format that distributes the points to make the network easier to view. Planned utilization, link vendors and other attributes can be viewed by color; hardware devices are represented by various icons. From the topology map, popup menus can be accessed by right-clicking on nodes or links, providing quick targeted access to information about the selected network element.

Add-On Features

Feature	Description
Discrete Event Simulation	Packet-by-packet simulation is available for Frame Relay and Router device models, giving statistics on queueing delays and packet loss ratios.
Domain/Structured Networks	Partitioned or structured networks, including PNNI, are supported by IP/MPLSView. Device-specific support of domains is also correctly modeled.
Traffic Load Analysis	Real traffic load per demand can be loaded into the software model to obtain a more accurate link utilization for failure simulations. IP/MPLSView can do multi-period load and performance analyses.
Facility Feature	Any number of nodes and/or links can be grouped together in a facility for failure scenarios and reporting options.
Class of Service (CoS) Feature	The class of service feature allows the user to classify traffic into four classes, including the strict priority class. With this feature, the accuracy of the Design and Discrete Event Simulation modules are greatly improved, as IP/MPLSView takes into account all the details of Class-Based Weighted Fair Queueing (CBWFQ) in its routing process.
Customer Tariff	The Customer Tariff allows the user to specify various tariff classes and their rates for the network. Each node is associated with a particular tariff class. All links would then be priced according to its service and the classes of the nodes it connects.

Supported Hardware

Generic hardware devices include ATM, Frame Relay, IP router, optical transport, voice, and TDM. Supported devices and protocols are listed below, grouped by hardware vendor. The hardware vendors you can model in IP/MPLSView depend on your license file.

Alcatel

- ATM/7670/7470/5620 or PNNI-based routing
- 7750 Service Router

Cisco Systems

- Routers: IOS and IOS-XR-based devices such as 7500, GSR12000, and CRS-1
- MPLS/Tag Switching (MPLS-TE, GB-TE)
- PIX Firewall
- LightStream 1010
- IGX 8400, BPX 8600, and MGX 8800
- ONS 15454, 15327, 15000 (optical transport)

ECI Telecom LTD

- T::DAX

Force10

- Force10 Router

Foundry

- NetIron IMR, XMR Series

Huawei

- NE and AR Core Routers

IBM

- Nways 2220

Juniper Networks

- Routers: JUNOS and JUNOSe-based devices such as E- (including ERX 700, 1400), J-, M-, T-Series
- NetScreen Firewall

Lucent Technologies (Ascend Communications)

- B-STDx
- CBX 500, GX 550

Marconi

- ForeRunner ASX, TNX

net.com

- Micro20, IDNX20/70/90
- Promina 100/200/400/800
- FrameXpress, CellXpress, SCLX
- SCREAM 50/100
- STM

Nortel

- Multiservice Switch 6400/7400/15000/20000
- MPE 9000 Series

Riverstone

- RS Router Family

Sycamore

- SN3000, SN16000, SN16000SC

Tellabs

- 8800 Multi-Service Router Series

RAD

- Megaplex Series, IPmux Series

Converting Your Files to IP/MPLSView File Format

The following is a partial list of existing utilities that can be used to convert your files to IP/MPLSView file format. Please contact Juniper if the extraction utility you are looking for is not listed here or if you would like to request customized extraction utilities.

Extraction Tools for Topology and Provisioned Data

Hardware	Extraction Utility	Author
Router	getipconf	Juniper
Lucent ATM	cv2wandl	Juniper
Cisco ATM	NMT	Cisco
Alcatel ATM	al2wandl	Juniper
net.com	ncp	Juniper

Extraction Tools for Traffic Data

Hardware	Extraction Utility	Author
Router	getiptraf, conviptraf	Juniper
Lucent ATM	bulk2obj, obj2wandl	Juniper

Documentation Conventions

Following are the documentation conventions used in IP/MPLSView.

Document Conventions

- Keyboard keys are represented by bold font in brackets; for example **<Enter>**.
- Window titles, field names, menu name, menu options, and Graphical User Interface buttons are represented by a bold font.
- Command line text is indicated by the use of a `constant width type`.

Keyboard, Window, and Mouse Terminology and Functionality.



Informational Note: In the documentation, mouse button means left mouse button unless otherwise stated.

-
- **Window.** Any framed screen that appears on the interface.
 - **Cursor.** The symbol marking the mouse position that appears on the workstation interface. The cursor symbol changes; for example, in most cases, it is represented as an arrow; in a user-input field, the cursor symbol is represented as a vertical bar.
 - **Click.** Refers to single clicking (pressing and releasing) a mouse button. Used to select (highlight) items in a list, or to press a button in a window.
 - **Double-click.** Refers to two, quick clicks of a mouse button.
 - **Highlight.** The reverse-video appearance of an item when selected (via a mouse click).
 - **Pop-up menu.** The menu displayed when right-clicking in or on a specific area of a window. Move the cursor and click the mouse button to make a selection.
 - **Pull-down menu.** The Main Window window menus on the tool bar. Move the cursor and click the mouse button to make a selection.
 - **Radio button.** An indented or out-dented button that darkens when selected.
 - **Check box.** A square box inside of which you click to alternately check or uncheck the box; a check mark symbol is displayed inside the box when it is “checked.” The check mark symbol disappears when the box is “unchecked.”
 - **Navigation.** When you type text into a field, use the **<Tab>** key or the mouse to move to the next logical field. Click inside a field using the mouse to move directly to that field.
 - **Gray or Grayed-out.** A button or menu selection is described as gray or grayed-out when it is inactive or inaccessible.

The Keyboard

The cursor keys located on the lower two rows of this keypad perform cursor movement functions for the window cursor. They are labeled with four directional arrows on the key caps. IP/MPLSView makes use of these keys for cursor movement within files.

The following keys or key combinations can be used in the except where noted:

- Click on a file then hold down the **<Shift>** key while clicking on another file to select the file first clicked on and all files in between.
- Click on a file and then hold down the **<Ctrl>** key while clicking on another file to select the file first clicked on and the file next clicked on without selecting any of the files in between. You can continue to **<Ctrl>**-click to select additional, single files.

The Mouse

The following terms describe operations that can be performed with the mouse.

- **Point.** Position a mouse pointer (cursor) on an object.
- **Click.** Quickly press and release the left mouse button without moving the mouse pointer.
- **Right-click.** Quickly press and release the right mouse button without moving the mouse pointer.
- **Double-click.** Quickly click a mouse button twice in succession without moving the mouse pointer.
- **Press.** Hold down the mouse button. (Unless otherwise specified, the left mouse button is implied.)
- **Release.** Release a mouse button after it has been pressed.
- **Drag.** Move the mouse while a mouse button is pressed and an item is selected.

Informational Notes, Cautions, and Warnings

Informational notes are special notes placed in a document to alert you of an important point. This document makes use of the following “Informational Note” label:



Informational Note: This is an informational note.

Instances where there is the possibility of inconvenience, temporary loss of functionality, unwanted data loss, or other undesirable outcomes are indicated as Caution notes.



Caution: This is a caution-type note.

A Warning note indicates very important information which needs to be followed to avoid risk of permanent hardware or software damage or personal injury.



Warning: This is a warning-type note.

Changing the Size of a Window

You can change the size of windows (with some exceptions such as dialog boxes) by pointing to a border or corner of the window's frame, pressing the left mouse button, and dragging the window's frame until the window has reached the desired size. You also can click on the minimize, maximize, and exit buttons in the upper right-hand portion of the window:

Moving a Window

You can move a window by pressing your mouse down when your pointer is on a window's top border. Keep your mouse's left button pressed down and drag the selected window to the place of your choice. When you are satisfied, release the mouse button.

For Further Information

For more detailed information, refer to the subsequent chapters in this document and to the IP/MPLSView Java-Based Graphical User Interface Reference.

Chapter 3

Understanding the Specification File

This chapter explains what a specification file is and how it is used to organize the network files that make up each IP/MPLSView network project.

To import router configuration files to automatically create a specification file, see the Router Feature Guide for IP/MPLS chapter on router data extraction. For information on how to create a paper model through the client, see Chapter 5, Creating and Editing a Network Model.

Prerequisites

Your server and client should be running as described in the Getting Started Guide for IP/MPLSView.

Related Documentation

For a more in-depth look at the input file formats, refer to the File Format Reference for IP/MPLSView.

For extraction utilities that can automatically create the specification file from network raw outputs, such as config files, refer to the hardware-specific guide or contact Juniper support.

Definition of a Specification File

The specification file, short for “specification” file, is the file that indicates the file locations of IP/MPLSView input files. When loading a network, as described in Chapter 4, Opening and Closing a Network Project, the specification file will be used to find the information needed to construct the network model. Here is an example specification file:

```
Filename:spec.y
runcode=x
datadir = /u/wandl/data/example
dparam=dparam.x
muxloc = muxloc.x
nodeparam = nodeparam.x
bblink = bblink.x
demand = demand.x
```

Specification File Format

- Each entry of the specification File is a line of the following general format:
`<keyword>=<value>`
- File locations should be indicated using the format: `<file type> = <file path>`

-
- Directory locations are indicated using the format: *<directory type> = <dir path>*

runcode

The **runcode** is a user-specified file extension that enables you to quickly identify the network files belonging to the same project. In the above example, the runcode is 'x' and all the files have 'x' as the file extension. Choose a runcode that will help you to distinguish between different versions of a network model that you are studying. For example you could distinguish three separate network designs using runcodes "1", "2", and "3".

datadir

The **datadir** is the default directory where the input files are located. In the above example, the input files will be searched in /u/wandl/data/example. The default directory can be overwritten in file location specifications by specifying an absolute path for the file location.

File Types

This specification file indicates the location for separate files with information about nodes, links, and demands. Each of these files has a special keyword to indicate the file type. In the above example, **muxloc** is a file type indicating node location information. The **nodeparam** file is a file type indicating additional node parameters. The **bblink** file provides backbone link information including the node endpoints. The **demand** file provides planned traffic information including the source, destination, and bandwidth information. For a more complete list of file types, refer to the File Format Reference for IP/MPLSView.

File Names

In the above example, the muxloc, nodeparam, bblink, and demand files will be searched for in the respective files muxloc.x, nodeparam.x, bblink.x, and demand.x in the data directory /u/wandl/data/example.

Note that the files are normally organized in the same directory, the **datadir**. Additionally, they are named according to the file type and runcode, where the file type is the prefix and the runcode is the suffix: *<filetype>.<runcode>*

It is recommended to follow this convention to allow you to easily identify a network. The program will be using this naming convention when saving a network. For example, suppose you save the above example using directory /u/wandl/data/y and runcode y. Then the files muxloc.y, nodeparam.y, bblink.y, and demand.y will be created in /u/wandl/data/y.

Creating Input Files

This section describes how to create input files by hand. See Chapter 5, Creating and Editing a Network Model for instructions to create a specification file using the graphical user interface. Refer to your hardware-specific guide or the IP/MPLSView Java-Based Graphical User Interface Reference for information about extraction utilities to automatically convert raw data collected from your network to IP/MPLSView file format.

The minimum information needed in the specification file is the runcode or file extension and the location of a *dparam* file (default parameters file) specifying the hardware type. The following is an example of a minimum specification comprised of a specification file and dparam file.

```
Filename: spec.x
runcode=x
dparam=dparam.x
```

```
Filename: dparam.x
hwvendor=GENERIC
```



Informational Note: You should choose a hardware vendor that you have permission for in your license file.

In this example, the runcode is "x". With this basic specification file, you can open your empty network in the IP/MPLSView client by double-clicking on *spec.x* within the File Manager. You can then proceed to add nodes, links, and demands. When you save your design environment (**File > Save As**), select the directory destination to save the network and specify the runcode (file extension) of the network files. Saving to the current runcode will overwrite and update your existing network files. If you do not want to overwrite existing network files, specify a new runcode. Your new specification file will now contain more detailed file specifications for the newly added nodes, links, demands, and other information you added. The network input files can also be created using the editor in the **File Manager**. Example file templates can be found in the **File Manager** right-click menu's **New** sub menu.

Example Set of Network Files

```
Filename: spec.x
runcode=x
dparam=dparam.x
muxloc=muxloc.x
nodeparam=nodeparam.x
bblink=bblink.x
dparam=dparam.x

Filename: dparam.x
hwvendor=GENERIC

Filename: muxloc.y
ATL ATLANTA 404 72
BOS BOSTON 617 661
CHI CHICAGO 312 899

Filename: nodeparam.x
ATL  NODE
BOS  NODE
CHI  NODE

Filename: bblink.x
LINK1 ATL BOS DEF 1 OC3 DIST=100
LINK2 BOS CHI DEF 1 OC3 DIST=100
LINK3 CHI ATL DEF 1 OC3 DIST=100

Filename: demand.x
DMD1 ATL BOS 730017 R,A2Z 02,02 ATL--BOS
```

For help on the file formats, or the keywords required in the specification file, see the File Format Reference for IP/MPLSView.



Informational Note: Some users find it easier and faster to edit the text files directly, especially when creating the initial network. Others may find it easier to use the graphical interface and have the network files and specification file automatically generated by the program. For more on how to create a new network project via the graphical interface, see Chapter 5, Creating and Editing a Network Model.

Adding Comments

You can add comments to your network files by using the “#” (pound) sign. These line entries will be ignored. For example, comments can be used to list the headings for the data columns in the muxloc (node) file:

```
#ID NAME NPA NXX
```

However if the program overwrites your existing network files (via **File > Save**), not all comments will be preserved. To preserve comments use a pound sign followed by an exclamation mark (for example, ‘#!’). Note this only preserves those comments that are at the end of a valid entry. For example, suppose comments are added to the muxloc (node) file as follows:

muxloc.y Before Save	muxloc.y After Save
#comment1	ATL ATLANTA 404 724
#!comment2	BOS BOSTON 617 661 #! comment4
ATL ATLANTA 404 724 #comment3	
BOS BOSTON 617 661 #! comment4	
CHI #! comment5	

As shown in the above example, upon saving the file, only lines containing valid entries were saved. And among those valid entries, only comments preceded by a ‘#!’ are saved.

Additional Notes

Once the node, link, and demand files are created, they could be referenced from more than one specification file. Note that a change to an input file will impact each of the specification files referencing it. To avoid changes to a shared input file, the owner of these files can set the file write permissions. For example, to make a file read-only, enter the following Unix commands: “cd *specfile_directory*” and “chmod 444 *filename*” where *filename* is substituted by the name of the file to be set as read-only.

datadir - Specifying the Location of Input Files

As mentioned earlier, the keyword “datadir” can be used in your specification file to specify the default directory that input files (for example, muxloc, bblink, dparam, and demand files) will be located in. The datadir is provided simply for convenience, so that you do not have to type out the full path for each of the input files. By default, the datadir is the directory containing the specification file. For any input file, either a relative path should be specified from the datadir or an absolute path should be specified that will override the datadir.

For example, suppose there are two users, amy and bob, in the same user group, with access to the files muxloc.amy, bblink.amy, and demand.amy in /u/wandl/amy. For user amy, a specification file created in /u/wandl/amy may reference those files as follows:

```
Filename: /u/wandl/amy/spec.amy
runcode=amy
#datadir = /u/wandl/amy
muxloc = muxloc.amy
bbblink = bblink.amy
demand = demand.amy
```

For user bob, he may create a specification file in /u/wandl/bob referencing the input files from /u/wandl/amy by copying only the file spec.amy and removing the comment in front of the datadir. Because the input files are in a different directory as the specification file, Bob can either specify their absolute path (see example at left) or specify a datadir containing the input files (example at right). Both of the following specification files in the following table are equally valid but the right one is more concise. If the data dir is omitted in this case, the default directory will be /u/wandl/bob and the files will not be found.

/u/wandl/bob/spec.bob (using default datadir /u/wandl/bob/)	/u/wandl/bob/spec.bob (specifying datadir /u/wandl/amy)
runcode=bob2 muxloc = /u/wandl/amy/muxloc.amy bbblink = /u/wandl/amy/bblink.amy demand = /u/wandl/amy/demand.amy	runcode = bob datadir = /u/wandl/amy muxloc = muxloc.amy bbblink = bblink.amy demand = demand.amy

Later, Bob may decide that he wants to use his own link file in his own directory. In that case he can change the bblink file path as follows:

```
Filename: /u/wandl/bob/spec.bob
runcode=bob3
datadir = /u/wandl/amy
muxloc = muxloc.amy
bbblink = /u/wandl/bob/bblink.divdesign    # changed bblink file
demand = demand.amy
```

Making a Copy of a Specification File

The easiest way to make a copy of a set of network files is to open the specification file and then save it using a different runcode and/or a different directory from **File > Save As**.

Opening and Closing a Network Project

This chapter describes how to open an existing network project file in the IP/MPLSView client. As described in the previous chapter, each specification file, or “specification” file, represents a different network project, and contains references to all the individual network files that comprise the network project.

When to use

Use these procedures to open a network project file that has been previously created.

Prerequisites

You should already have a basic understanding of specification files, as described in Chapter 3, Understanding the Specification File. If you have not yet created your own specification file, you can open one of the existing specification files located in the IP/MPLSView sample directory. If you wish to create your own specification file, see Chapter 5, Creating and Editing a Network Model.

You should have already logged into the client as described in the Getting Started Guide for IP/MPLSView.

Related Documentation

For an overview of the **File Manager**, you can also refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Recommended Instructions

Following is a high-level, sequential outline of the process used to open a specification File and the associated, recommended procedures.

1. Open a specification file as described in Open Network Project File on page 17.
2. Close a specification file as described in Close Network Project File on page 19.
3. Re-open a specification file as described in Accessing Recently Opened Project Files on page 19.

Detailed Procedures

Open Network Project File

1. Use the **File Manager** window to navigate to the directory where the specification File is located.

-
- **Filtering Tip:** To view only files that are specification Files, you can select “Spec Files” from the **Files of Type** drop-down menu in the File Manager. Or, in the same field, type in a filter string (for example, “spec.*”) and press <Enter>. To restore the original file listing, remember to either choose “All Files” from the **Files of Type** drop-down menu, or clear this field and press <Enter>.
 - **Navigation Tip:** There are multiple ways to navigate. To go directly to a directory, you can type in the directory path in the browser bar and press <Enter>. Note that this field is case-sensitive. Alternatively, you can use the navigation tree in the left pane of the **File Manager**. Or you can select a recently visited directory by clicking on the arrow to the right of the browser bar.
2. Select the specification file you wish to open. For this tutorial, the MPLS-Fish router sample network will be used. By default, it is located in `/u/wandl/sample/IP/fish/spec.mpls-fish`.



Informational Note: If you have an ATM license as opposed to a ROUTER license, you can try opening `/u/wandl/sample/original/atm/uk/spec.uk`. If you encounter an error when trying to open the specification file, open the `dparam.uk` (default parameters) file, and search for the Hardware Related Parameter line that says “hwvndor = GENATM”. You may not have the Generic ATM hardware vendor in your license. Simply replace “GENATM” with a hardware vendor listed in your license (located in `/u/wandl/db/sys/npatpw`) and save the `dparam` file. Now you should be able to open the specification file.

3. Double-click the specification file.

Informational messages will scroll on the Console window while the network files are being loaded. Check these messages for input file integrity check errors. You may be prompted with the following dialog box shown below. This dialog box normally shows up for router networks.

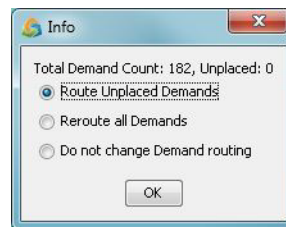


Figure 1: Update Demand Paths

- Selecting “**Do not change Demand routing**” will cause the IP/MPLSView to route the end-to-end demands according to any paths already defined in the demand file.
- Selecting “**Route Unplaced Demands**” will cause unplaced demand paths to be placed from scratch. All other demands will be routed according to their previously defined routes.
- Selecting “**Route all Demands**” will cause all demand paths to be placed from scratch.



Informational Note: The path placement can always be updated later from the menu **Design > Demands > Route Paths**

- Click **OK**. The network will then be displayed.

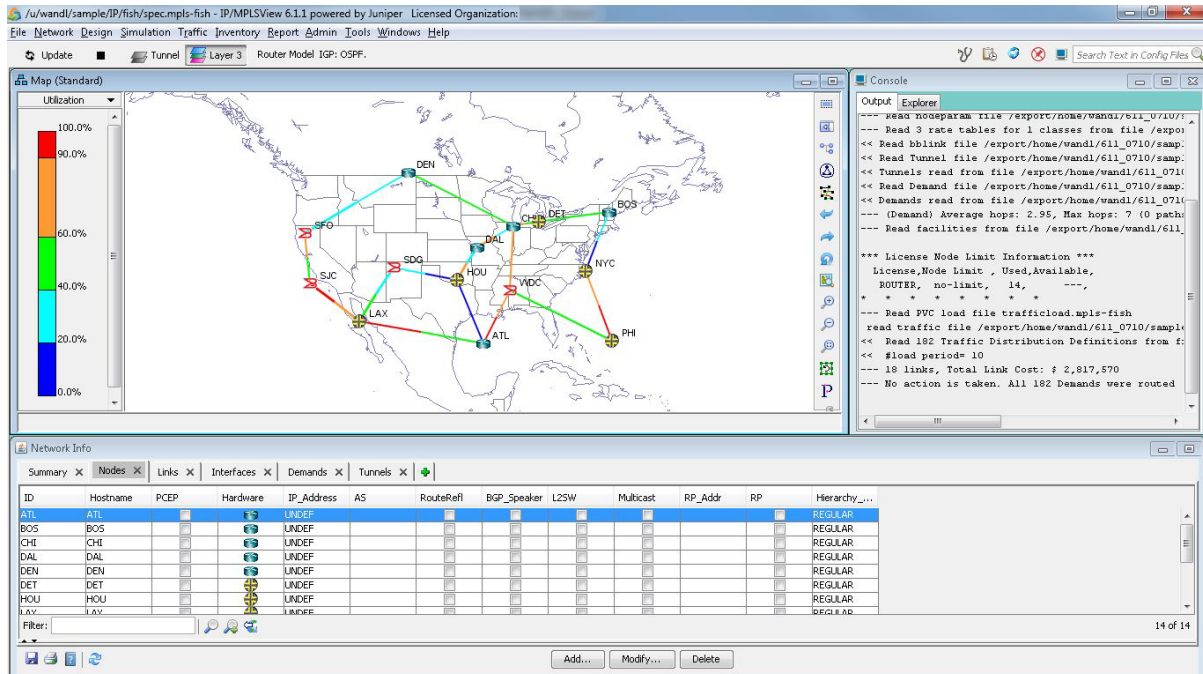


Figure 2: Topology Window Upon Opening MPLS-Fish Specification File

Two subdirectories will be created under the project directory: Log and Report.

- The **Log** directory is used to save error messages from parsing of input files such as demand file, tunnel file, path tables and tunnel path files.
- The **Report** directory is used to save all the reports created by rtserver from simulation and from Report menus. This report directory can be changed by specifying `reportdir=xxx` in `dparam` file.

Close Network Project File

From the main menu, select **File > Close** to close the project.



Informational Note: If modifications were made, you may be asked whether or not you want to save these changes. In this case a dialog box will be displayed to specify the directory and runcode to save the project. If you save the project under the same runcode in the same directory, this overwrites previous files. Alternatively, you can save the project under a different runcode or in a different location.

Accessing Recently Opened Project Files

Recently opened project files are listed under the **File** menu or in the **Network Browser** Recently Opened tab.

Creating and Editing a Network Model

This chapter explains how to create a new paper network model via the IP/MPLSView client. You may create a new network model from scratch, or you may load parameters from a previous network model.

Note that these steps are not necessary if you have existing router configuration files. To import router configuration files to automatically create the network model, see the Router Feature Guide for IP/MPLS chapter, “Router Data Extraction.”

Definition of a Specification File

Each specification file correlates to a network project. As described in Chapter 3, Understanding the Specification File, the specification file is a file that specifies where to look for different kinds of information regarding the particular network model. Such information includes information about nodes, links, demands, and more.

If you are starting a network from scratch, the minimum information needed in the specification file is the dparam file and the runcode (system file name extension). Be sure to specify the hardware type in the dparam file. The runcode is used to identify the network files that belong to the same network project. Whenever you save your network environment, all the files associated with your network will be given the same runcode, or file name extension, of your choosing.

When to use

Use the procedures outlined in this chapter to create a specification file for your model network using the IP/MPLSView client. Alternatively, to create a specification file using the text mode, see Chapter 20, Text Mode. If network files in IP/MPLSView format have already been created, their locations can be specified in the specification file during specification file creation.

Recommended Instructions

The following is a high-level sequential outline of the specification file creation process.

1. Create a new directory for a specification file.
2. Create a specification file in that directory.
3. Edit a specification file after it is created.
4. Create a network file using the text editor templates and edit the specification file accordingly.

Detailed Procedures

The following section describes how to create and edit a specification file.

Creating a Directory for Your Network

1. Log onto the IP/MPLSView client and open **File Manager** from the File menu.
2. In the **File Manager** window, navigate to the data directory where IP/MPLSView is installed (example, /u/wandl/data)
3. To create a directory for your network files, right-click on the right pane of the **File Manager** and select **New > Directory**.
4. The **New Directory** pop-up window will appear. Specify a name for the directory and then click **OK**.

Create a Specification File

1. Locate the new directory you created and double-click the folder icon.
2. Right-click the right pane of the **File Manager** window and select **Spec File > New Spec**. Alternatively, from the main menu select **File > Create Network > Paper Model**. This opens the Spec File Generation Window.

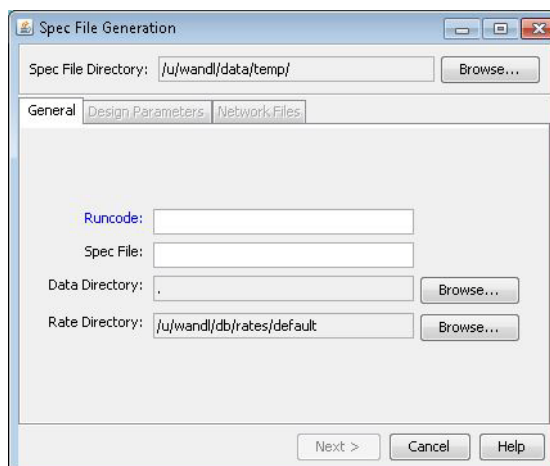


Figure 3: Spec File Generation Window

3. The **Spec File Directory** is the directory that your specification file will be saved to. The directory can be changed by clicking the **Browse** button.
4. Enter a **Runcode** for your network project. The specification file will use this extension.
5. The **Data Directory** is set to '.' by default which is the same directory as the specification file directory. The directory can be changed by clicking the **Browse** button.
6. The **Rate Directory** is the directory where the tariff database files are stored. Normally the default path does not need to be changed unless users wish to use their own tariffs.

7. Click the **Next** button. The **Design Parameters** tab is now active.

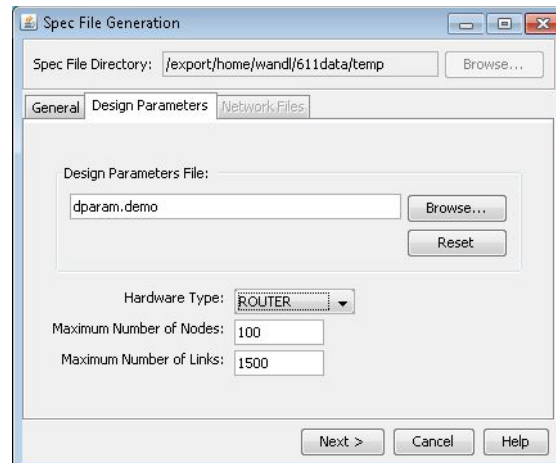


Figure 4: Design Parameters

8. First, select a **Hardware Type** from the drop down box. The available options will vary according to your license.
9. Second, set the **Max Node** and **Max Link** parameters for your network model or use the default values. These two parameters are used by the program to allocate memory efficiently and can only be changed when a network is closed. A good rule of thumb is to set the numbers according to the planned number of nodes and links in the network, leaving some room for growth. The **Max Node** parameter should not exceed the node limit of your license file. Setting these values too high may use up too much memory. For the **Max Link** parameter, no additional links will be added during a backbone design run when the **Max Link** is reached.
10. Click on the **Next** button. The **Network Files** tab is now active. If you have existing network data files in IP/MPLSView format that you want to reference, continue to the next step. Otherwise, click on the **Finish** button to proceed.

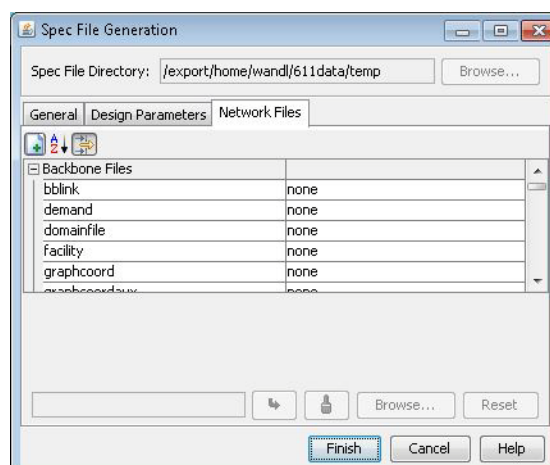


Figure 5: Network Files

11. Locate the file type you want to use in your network model. Select the row for the file you want to reference.

12. For the selected row, there are two ways to specify the chosen file. You can type the path of the file in the text box at the bottom of the window. Or you can click on the **Browse** button to navigate to that file.
13. If you want to undo the file reference, click the brush icon.
14. Click on the **Finish** button to proceed.
15. The **Spec Status** window will be displayed.

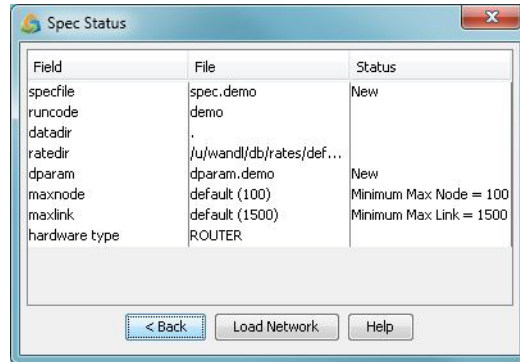


Figure 6: Spec Status Window



Informational Note: In the **Status** column, you will be able to see if your file is new or if it will overwrite an existing file. Another status you may see is **Missing** which indicates that a network file referenced was not found. Click on the **Back** button to make changes.

16. Click on the **Load Network** button to generate the network model. You may answer any dialog boxes with **Yes**. The Map window will open although there may be no elements at this time.
17. You can now proceed to add nodes, links, and demands to your network.

Editing a Specification File

After creating your specification file, you can edit it by right-clicking on the it from the **File Manager** window and select **Spec File > Modify Spec**. This will open the Modify Spec window.

If you're familiar with the specification file entries, you may right-click on the specification file and select **Edit** to change the entries in the text editor.

Creating or Editing a Network File via the Text Editor

This example will use file editing and file creation via the text editor to add a network file.

1. Use a previously created specification file. Navigate to the specification file's containing folder.
2. Create a network file (backbone link) in IP/MPLSView file format. This can be done from the **File Manager** by right-clicking the right pane and select **New > Backbone Data > bblink**. Enter in the filename using the same runcode extension as your specification file. This will provide a template for creating links.
3. Add an entry for one link:

```
LINK1  ATL  BOS  DEF  1  OC3
```


This will create 1 OC3 link named "LINK1" connecting nodes ATL and BOS with a default vendor type. Save and close the text editor.

4. Create a network file (mux location) in IP/MPLSView file format. This can be done from the **File Manager** by right-clicking the right pane and select **New > Backbone Data > muxloc**. Enter in the filename using the same runcode extension as your specification file. This will provide a template for creating nodes.
5. Add these two entries:

```
ATL           ATLANTA 404 214  US 33.757999  -84.388000
BOS           BOSTON  339 202  US 42.355000  -71.054001
```

This will create nodes ATL and BOS at their respective coordinates. Save and close the text editor.

6. Right-click on the specification file and select **Edit**. This will open the specification file in the text editor.
7. Add these two lines:

```
bbblink=bbblink.runcode
muxloc=muxloc.runcode
```

This will add references to the two newly created files. Save and close the text editor.

8. Double-click the specification file to load the modified network project.

The network model should display nodes ATL and BOS with an OC3 link connecting them.

Chapter 6

Loading a Network File

This chapter describes how to read or load in a network file into your network model. A file that is loaded in is not permanently a part of your specification file unless you save your design environment using **File > Save**.

When to use

This feature can be used to read in files, such as the backbone demand file, the link configuration file, the node weight file, and the user-defined link distance file. This feature can be used for traffic forecasting. It can also be used to change control files for different design runs.

Note that files specified in the specification file are read in automatically when the spec is opened. Files not defined in the specification files may be read in using **File > Load Network Files**.

Recommended Instructions

The following is a high-level outline of the load file process.

- Select **File > Load Network Files**.
- Go to the **Network Files** tab with the file parameter and read in the files.

Detailed Procedures

1. From the main menu select **File > Load Network Files**.

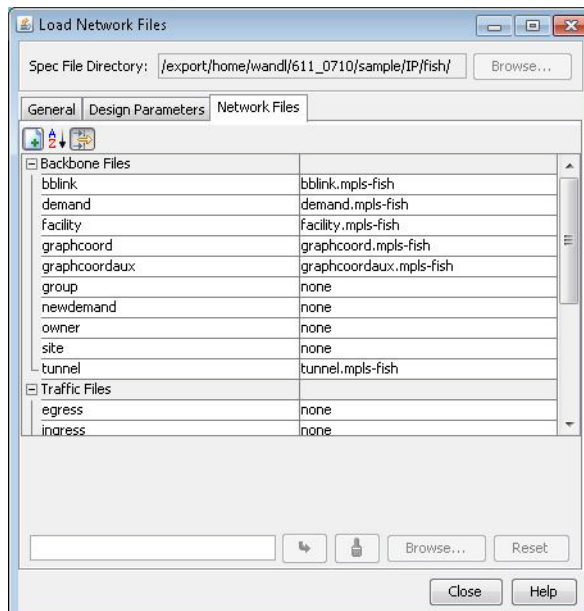


Figure 7: Load Network Files

2. Click on the **Network Files** tab and search for the file you want to update. Note that you cannot change the spec or dparam settings this way and their respective tabs will be grayed out.
3. Click on the row containing the file type you want to read. Click **Browse** to locate the file.
4. Click the **Load** (blue arrow) button to read the file.



Informational Note: The muxloc file is not selectable in this window. The muxloc file should be specified before loading the specification file. It may be done using the **File Manager**. To edit the specification file in the **File Manager**, right-click the file and select Edit.

Chapter 7

Using the Topology Window

This chapter describes how to display, work with, and customize the layout of the various topology windows.

When to use

Use these procedures to customize the graphical display and to access network information from the map.

Prerequisites

Open the sample/IP/fish/spec.mpls-fish network to follow along with the examples in this chapter.

Related Documentation

To learn how to open a specification file, see Chapter 4, *Opening and Closing a Network Project*.

For information on topology settings that are saved, and for details on additional topology features not discussed in this chapter, refer to the topology chapter in the *IP/MPLSView Java-Based Graphical User Interface Reference*.

Recommended Instructions

Following is a high-level outline of using the topology window.

- **Select and move Elements:** Select and rearrange nodes and links, undo and redo moves, and drag and rotate elements.
- **Network Information window and Finding Elements:** Viewing detailed element information. Find nodes and links.
- **Legends:** Learn how to use the left pane legends and change color settings in the legends.
- **Labels:** Label nodes and links in your network model, display the link distance and delay metrics, and hide the pricing message.
- **Geographical Map:** Display your model network superimposed against a geographical map.
- **Map Preferences:** Set your client map preferences, including color settings, font size, and link draw style.

-
- **Layout:** Set the network layout according to city latitude and longitude or arrange selected nodes using the Circle Nodes or Distribute Selected Points layout.
 - **Use the Navigator and Zoom Functions:** Use the navigator window and zoom in and out.
 - **Grouping:** Use the grouping functions.
 - **Protocol Subviews:** Choose subviews and use filter functions “&”, “or”, and “=” to view different logical combinations of subviews.
 - **Save Different Map Views:** Save and load different map views.
 - **Copy Layouts Between Maps:** Open a second topology window and copy the map layout from one map to another.
 - **Print, Export and Visio:** Print or export your topology map.

Select and Move Elements

Graphical vs. Geographical Coordinates

When you can select and move nodes on the Map this will change the *graphical* coordinates of the nodes, but the *geographical* coordinates will remain unchanged. The *graphical* coordinates can be saved with the Map View feature described later in this chapter. The *geographical* coordinates are saved in the muxloc file when you specify the latitude and longitude information of a node.

Element Selection

To select a single element, simply click on it. When one node is selected, its connecting links will and nodes will be highlighted. When one link is selected, its connected nodes will be highlighted.

To select several elements, <Ctrl>-click or <Shift>-click on each additional element. <Ctrl>-click or <Shift>-click on an element already selected will deselect that element.

To graphically select multiple elements, enable the Select Tool from the right-hand toolbar. Then hold down the mouse button to draw a target rectangle. When you let go, the nodes and links inside should be highlighted and selected. You can hold down <Ctrl> or <Shift> and continue to use the Select Tool to add more elements.

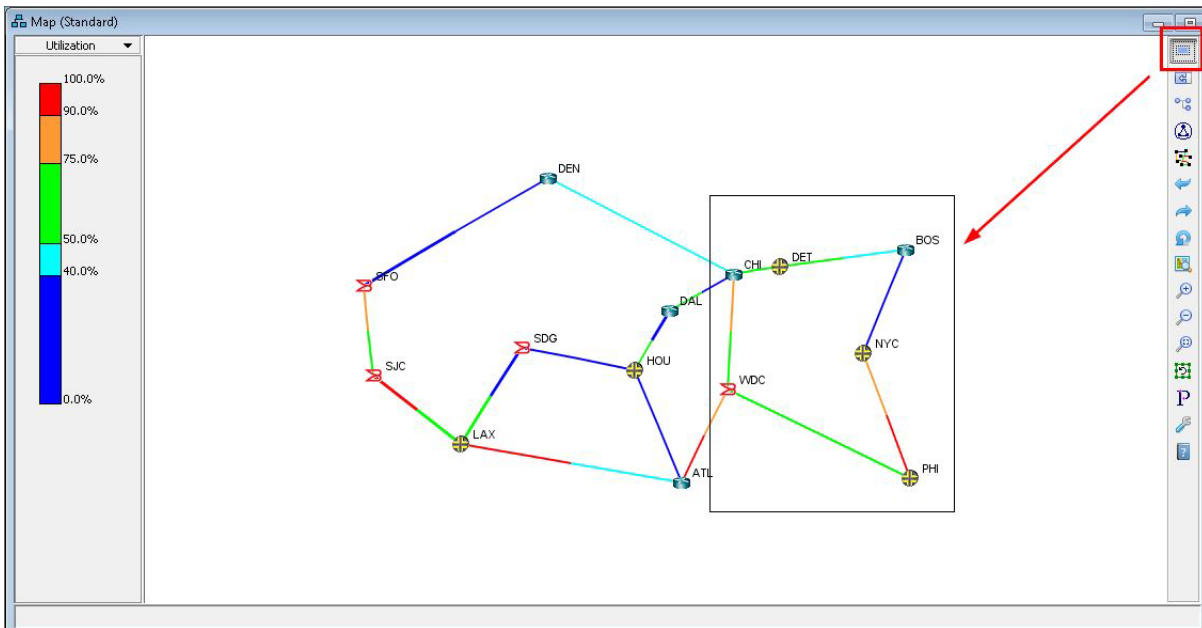


Figure 8: Selecting Elements Using the Select Tool

When elements are selected, you can move their positions on the map by using the mouse to drag and drop the elements.

To deselect all elements, click on any empty space in the Map.

Mouse click and hold on any empty space in the Map will allow you to drag and move the entire Map.

If you want to undo a move node action, click the **Undo Move Node(s)** icon.

If you want to redo a move node action, click the **Redo Move Node(s)** icon.



Figure 9: Undo and Redo Move Node(s) Icons

Using the Drag/Rotate Tool

The drag and rotate tool allows you to choose nodes or collapsed groups, and then stretch or shrink them, enlarge or decrease their area, move and rotate them.

Click on the drag and rotate tool. Then hold down the mouse button to draw a target rectangle.

A green box will be drawn with little square handles that indicates the area selected.

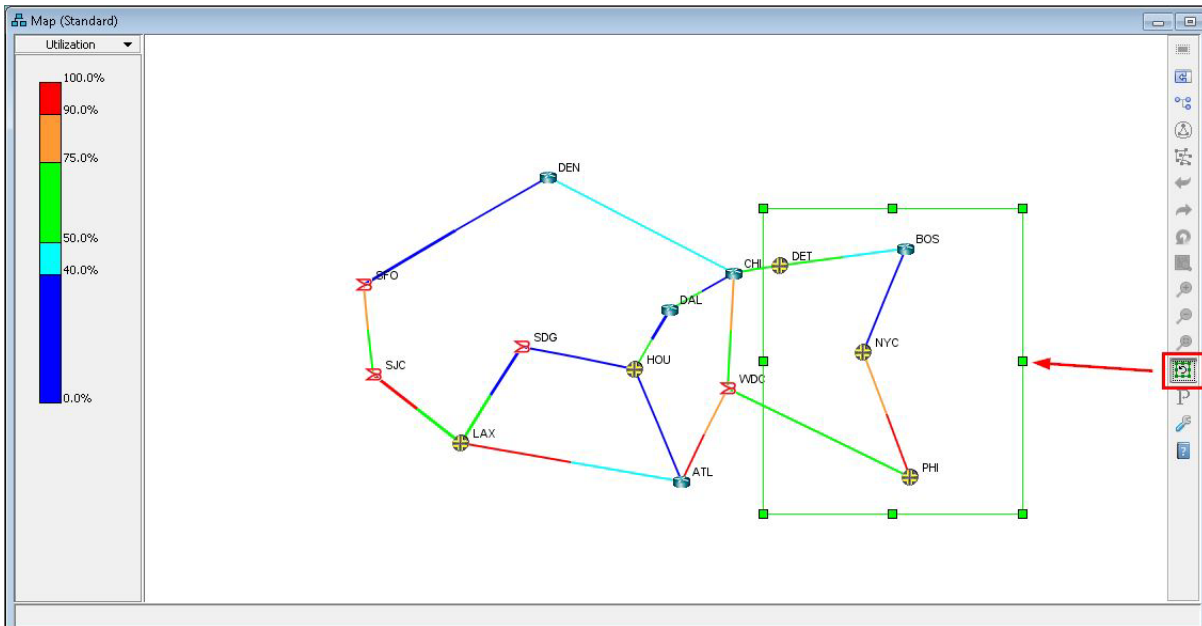


Figure 10: Drag and Rotate Tool Selection on the Map

Drag one of the eight square handles to stretch or shrink the region.

Drag inside the green box to move all the nodes inside.

Use the mouse wheel up or down to rotate the nodes. If you don't have a mouse wheel, click or <Ctrl>-click the middle mouse button to rotate the nodes.

To finish using the drag and rotate tool, click on any empty space in the Map.

Network Information Window

After selecting some nodes, right-click over a node and select **View > Selected Nodes**. Depending on what network element you click, the right-click menu may vary.

The information on the selected nodes is shown in the Network Info window Nodes tab.

You can see further detailed information of an element by toggling the up and down arrows to expand the view.

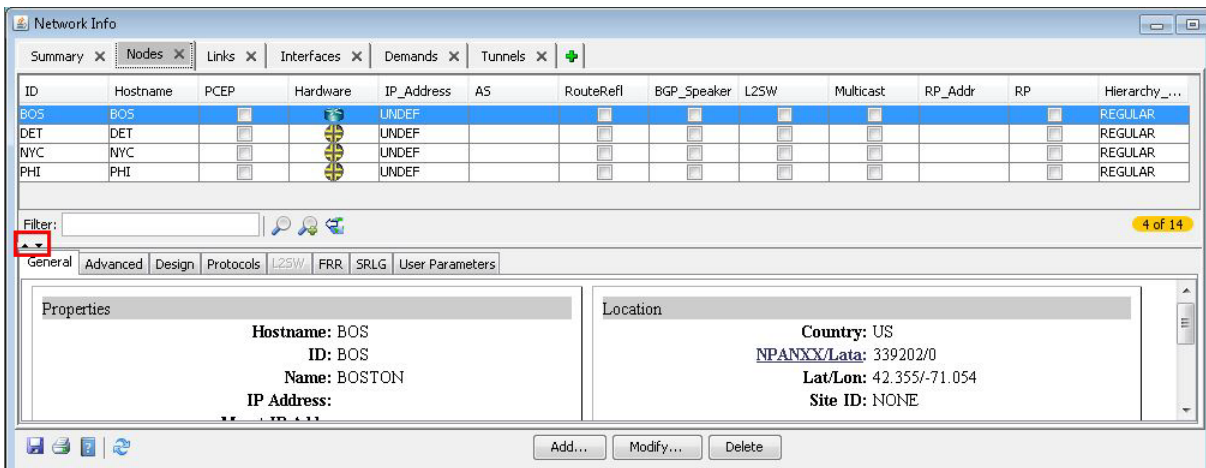


Figure 11: Network Info Window Nodes Tab

Finding Network Elements on the Map

To locate an element, right-click on the Map and select **Find** to open the search box.

Type in a node name and the associated node will be highlighted in the **Nodes List** in the left pane.

Type in an IP address and the associated link will be highlighted in the **Links List** in the left pane.

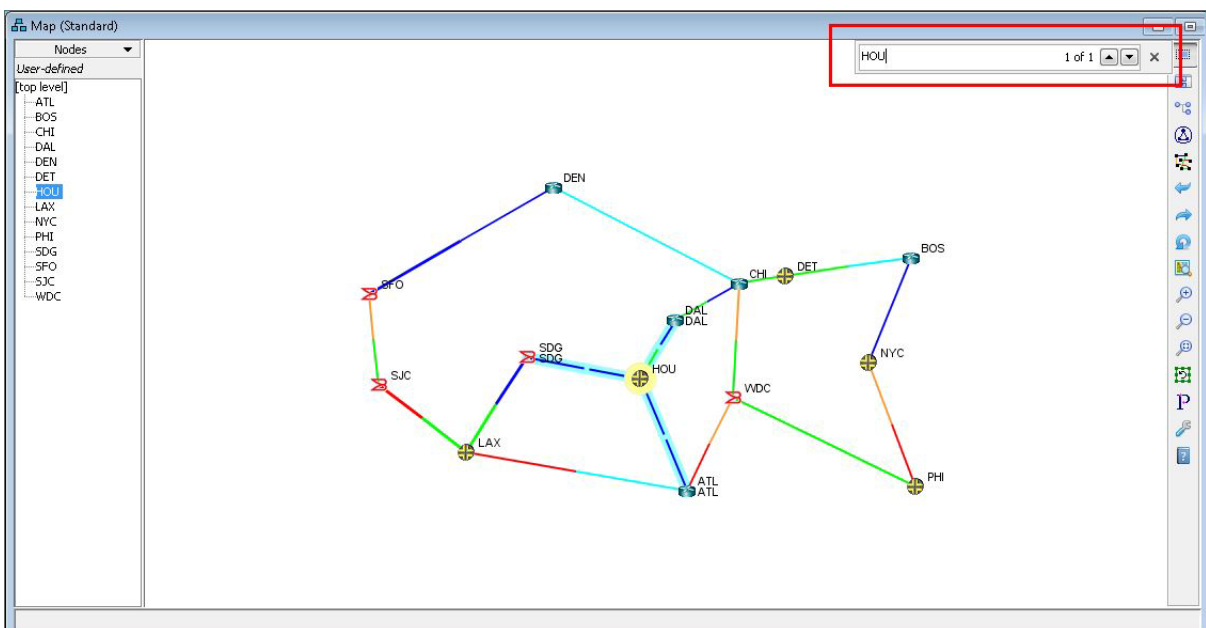


Figure 12: Find and Node List

Using the Legends

When the specification file is created or opened, the Map (Standard) window is displayed.

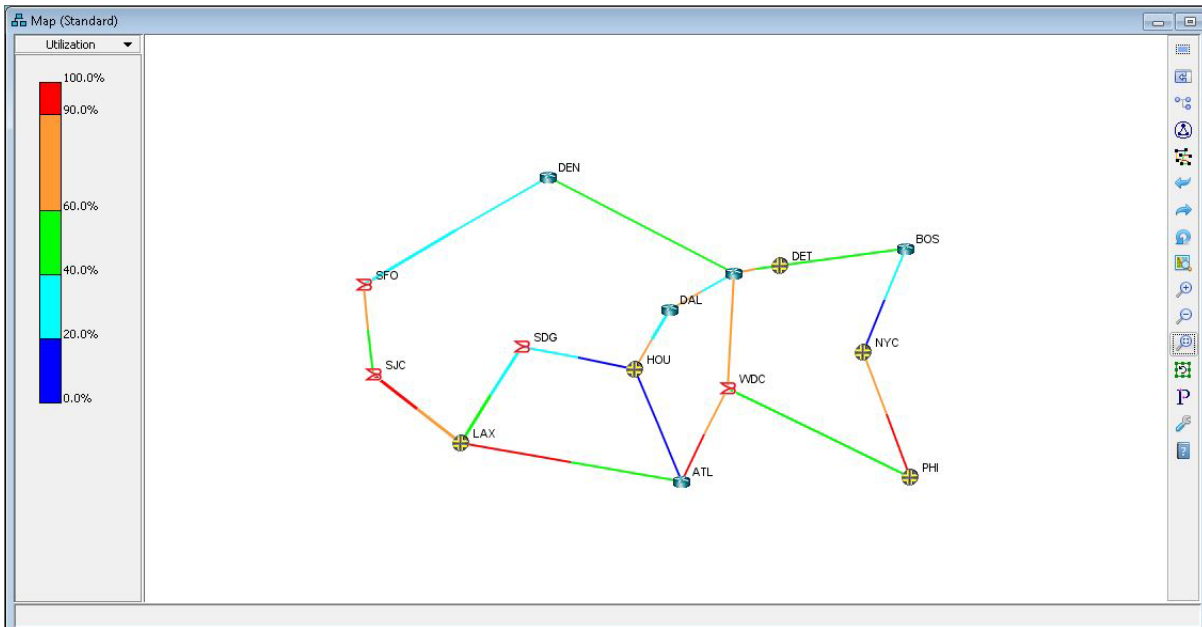


Figure 13: Map (Standard)

Utilization legend

Notice in the left pane there is a drop-down menu. By default, the **Utilization** legend is displayed and there are multiple legends listed in the drop-down menu. This legend displays the links' utilization by different colors. The colors and the utilization range can be changed.

Notice that some links are one continuous color while some links are two different colors at the mid point. The presence of two different colors indicates that the utilization in one direction (A->Z) is different from the utilization in the other direction (Z->A). The direction of the utilization color is read from the originating node.

On the **color bar**, drag the line separator between two colors up and down to move the separator and release it at the desired position. The number to the right of the separator indicates the utilization percentage corresponding to the selected position. For example, move the separator between the dark-blue segment and light-blue segment of the bar (currently at 25.0%) up to 40.0% and notice some formerly light-blue links are now dark-blue. The reason is because there were some links of utilization between 25.0% and 40.0%, and the color for that portion of the bar was changed.

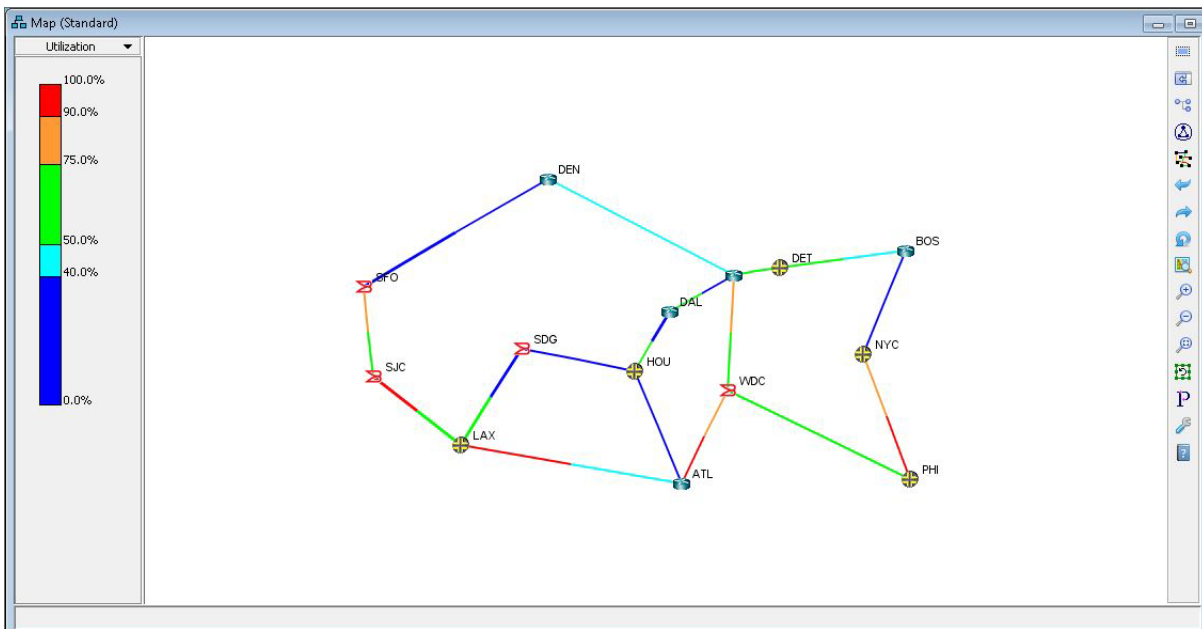


Figure 14: MPLS-Fish Network After Changing the Utilization Legend Settings

Right-click on the section between 90% and 100% to get a pop-up menu as shown below, and select the option **Edit Color**. The following **Edit Color** window will be displayed that will allow you to choose a different color for that section of the utilization bar.

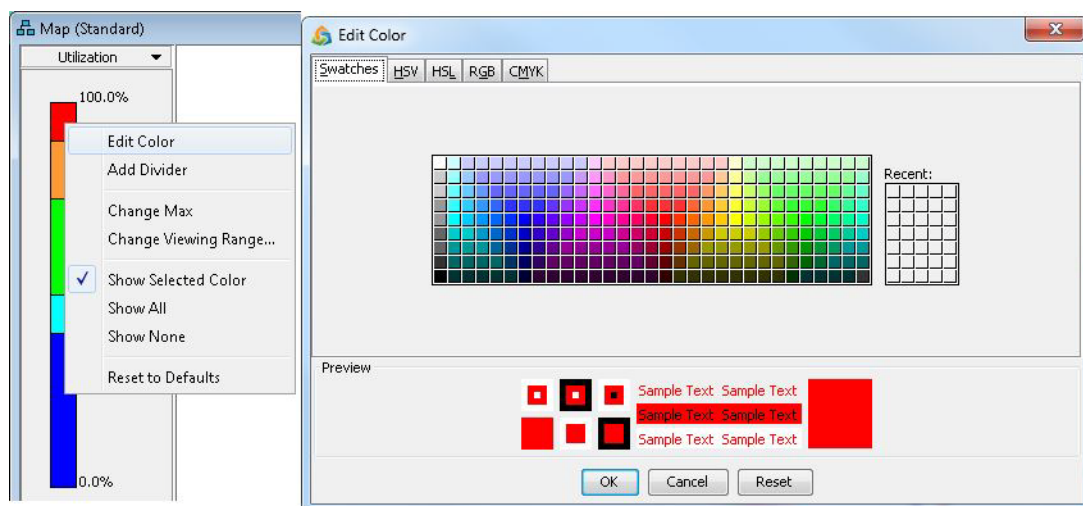


Figure 15: Edit Color Window

Select a color using any one of the options. The simplest way is just to click on a swatch of the desired color. For example, select purple. The **Preview** pane on the bottom of the window will give you an idea of how your color will look against other colors and text fields. Click **OK** to make changes.

Next, right-click on the color bar between 0% and 40%. Select **Add Divider** and you'll find that a new divider will be added at the place where you right-clicked. As with the other color dividers, you may use your mouse to drag and move the new divider up and down.

If there are links in the network model that are oversubscribed (utilization above 100%), you may want to see those specific links by color on the topology map. To do this, right-click on the color bar and select **Change Max**. A window will prompt you to enter a new maximum value for the color bar.



Informational Note: Settings in the **Legends** pane will be saved per specification file for future sessions. These settings are shared and applied among the four views: Utilization, Peak Utilization, Measured Link Utilization, and Demand CoS Utilization.

To display a specific link utilization range, right-click the color bar and select **Show None**. All of the links will disappear and the color bar will be grayed out. Then right-click the color bar in your desired range and select **Show Selected Color**. Any link within that utilization range will be shown on the map.

To restore all the links select **Show All**. A link of a particular utilization can also be hidden by toggling **Show Selected Color**. For links with two utilization colors, both utilization colors must be hidden for the entire link to be hidden.

To divide segments into smaller increments, right-click the color bar and select **Change Viewing Range**. Select a new viewing range. For example, 0.0 - 1.0 would be the range from 0% to 1%. This feature may be useful in cases where there are very small utilizations increments and finer granularity is needed.

Customizing Nodes and Links in the Legends

Other legends that you can use to examine your network: Filters, Network Elements, Utilization Legends, and Subviews. These are accessed from the map drop-down menu.

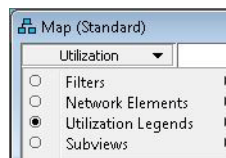


Figure 16: Map Drop-down Menu

Select **Subviews > Type** from the drop-down to see the different types of nodes and links in your network.

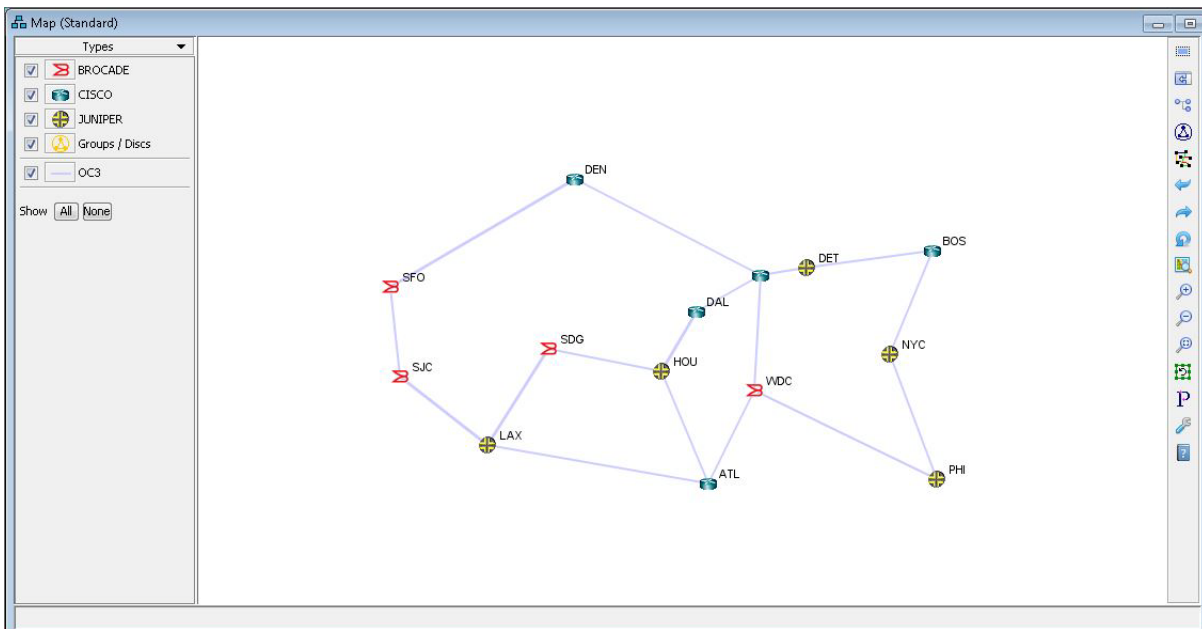


Figure 17: Types Legend

Select the drop-down menu a second time and notice that the **Subviews** sub-menu is now shown with the selected radio button on its left, and the items underneath it are provided as a shortcut to other menu items in the same category.

Each legend has its own color settings. Note that some legends change the link colors but leave the node colors the same as another previous legend. While some legends change the node colors but not the link colors.

Colors can be changed by clicking on the button next to the type of element you want to change.

Node icons and line styles can be changed by right-clicking over their respective buttons. For node icons use **Set This Icon**, and for link styles use **Set Line Style**.

Right-click a node or link button on the left pane and select **Highlight These Items** to highlight all nodes or links of a particular type on the Map.

Network Element Labels

Node Labels

To display node labels, right-click on the Map.

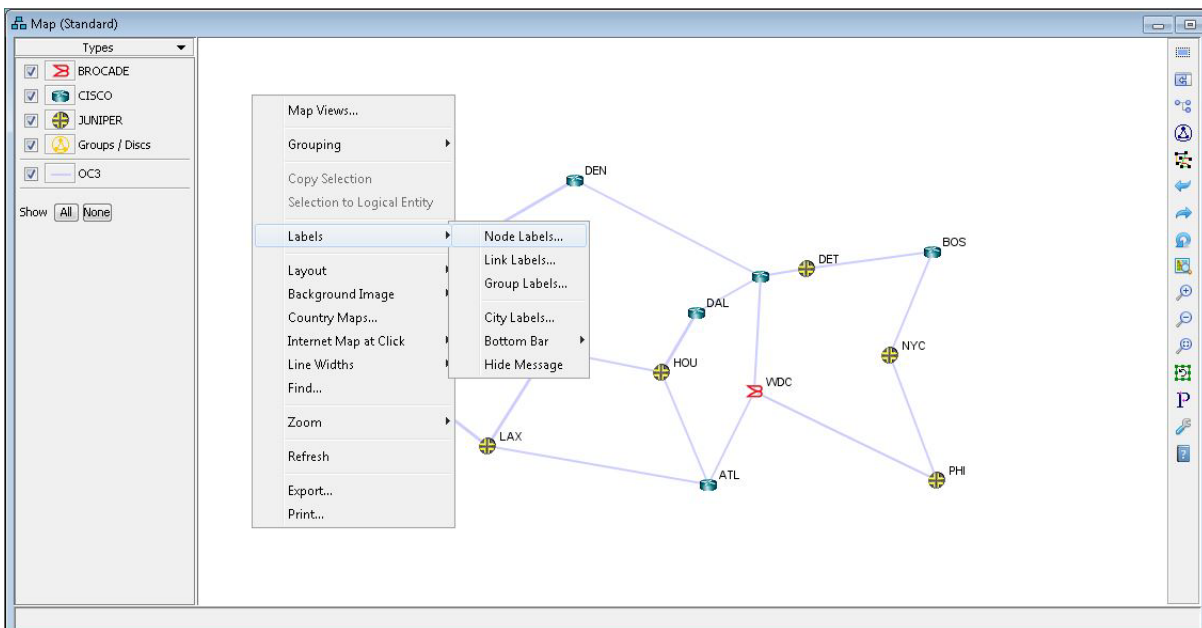


Figure 18: Right-Click Menu Labels Option

Select **Labels > Node Labels**. Choose **All** and **ID** then click **OK**. The network model will display the ID for all nodes.

You may also choose to only label selected nodes. To do so, first select the nodes you want to label, then right-click and go to **Labels > Node Labels**. Choose **Only Current Selection**.

Link Labels

To display link labels, right-click on the Map. Select **Labels > Link Labels**. Choose **All** and **Metric** to display the metric on all links.

You may also choose to only label selected links. To do so, first select the links you want to label, then right-click and go to **Labels > Link Labels**. Choose **Only Current Selection**.

When using label options that are dependent on Node A and Node Z attributes, such as Interface Name, the label name will be next to the associated Node. For example, if the map shows the position of NodeZ-NodeA, the Interface Name label will also be displayed as InterfaceZ-InterfaceA.

Bottom Bar Labels

It is possible to customize the information that appear on the bottom bar of the Map when you mouse-over a network element. Right-click on the Map and select **Labels > Bottom Bar**. You can customize the Nodes, Links, or Groups information to display.

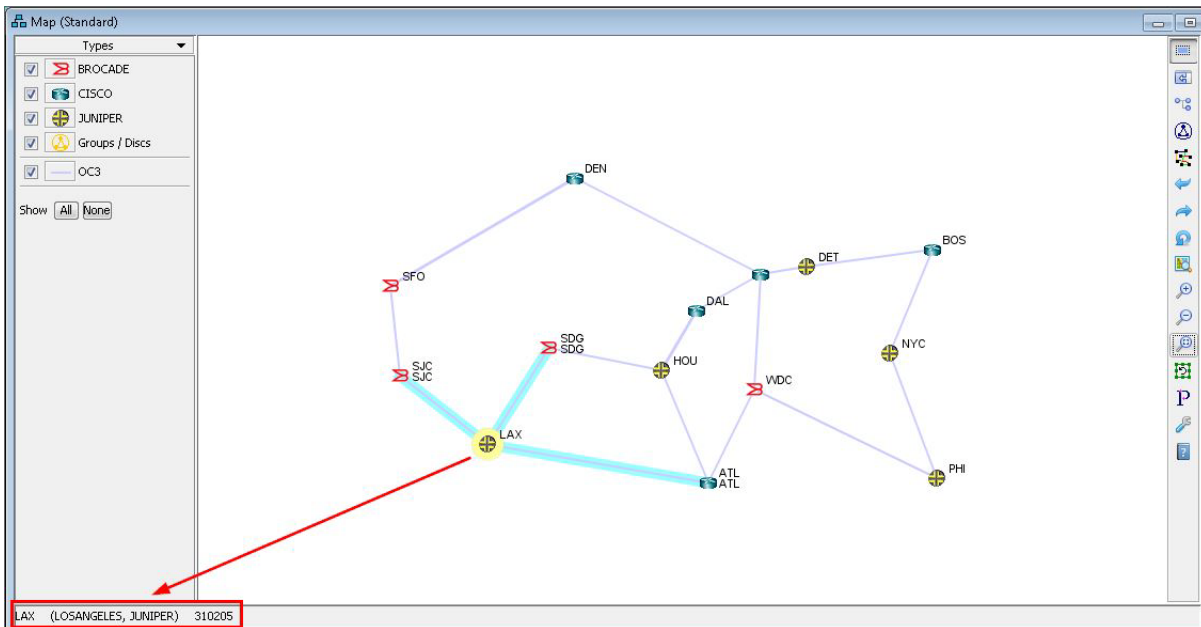


Figure 19: Bottom Bar Label

Displaying Geographical Maps

To see a country map, right-click on the Map and select **Country Maps**.

Choose the map option you wish to display. You can also specify how many cities to display on the map or latitude and longitude lines. Click **OK** to make changes.

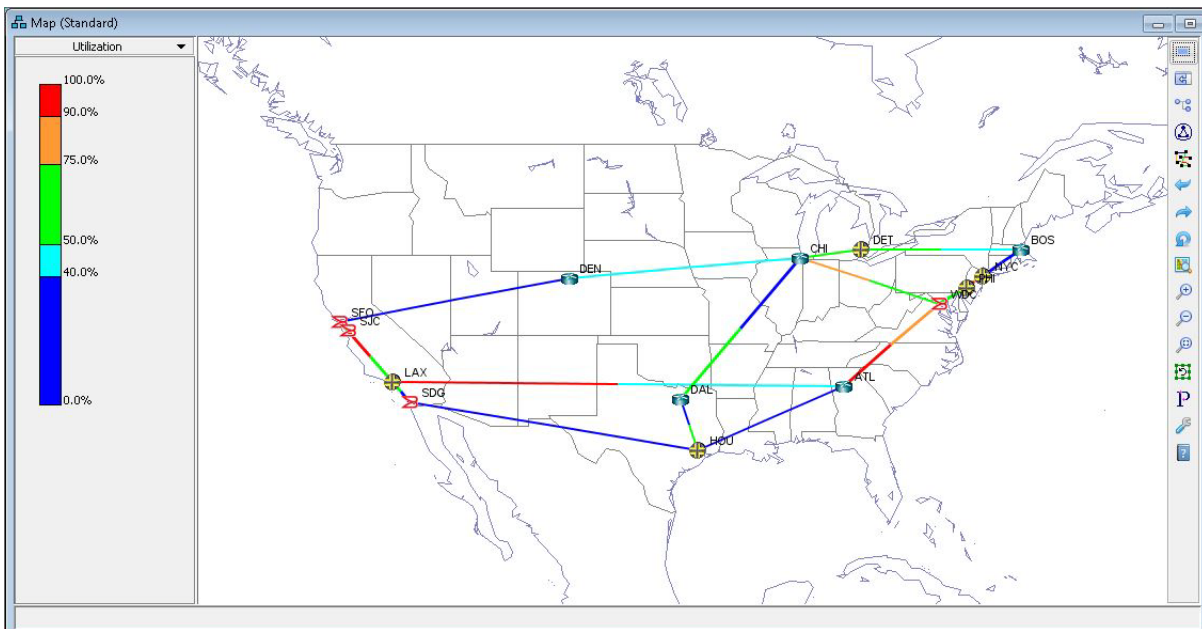


Figure 20: U.S. States Map Display



Informational Note: To arrange devices in their proper geographical locations, right-click on the Map and select **Layout > Reset by Lat/Lon**. This geographical layout only works if the coordinates are specified in the muxloc file.

Map Preferences

To change global color settings for your client, such as the background and foreground colors, from the main menu select **Tools > Options > Map Preferences**. The changes you make will be saved on your client machine.

The **Paint Discs** drop-down menu allows you to display groups using different paint settings.

Toggle the **Draw multiple links as curves** option to straighten out curved links.

Toggle the **Show utilization max instead of average within bundle** option to display the desired utilization when there are parallel links.

Toggle the **Animate Paths** option to view animated demand or tunnel paths.

Toggle the **Automatically save graphcoordaux file when closing a network** option to automatically save the position of node elements on the Map.

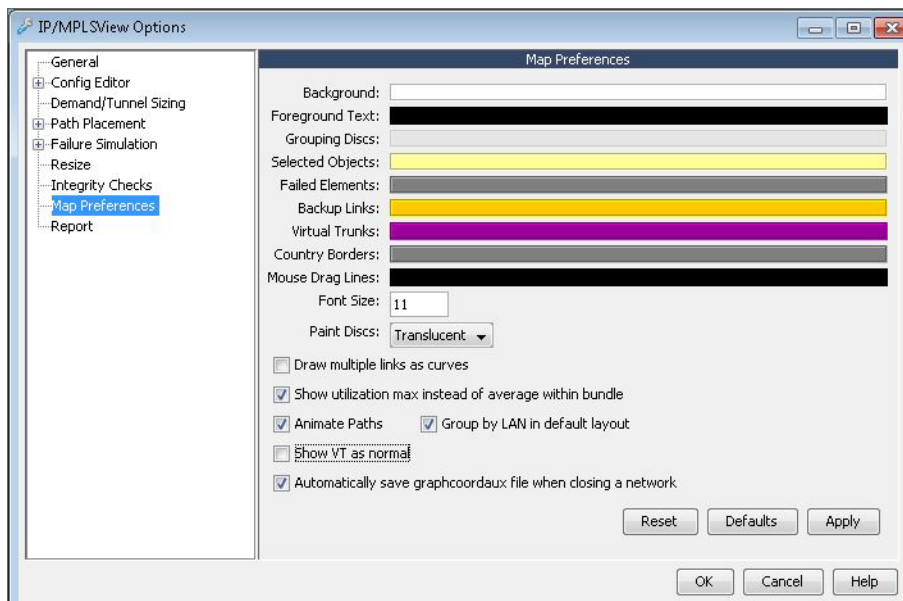


Figure 21: Map Preferences Window

Inverting Colors

Click on the white box of **Background** to change the Map background color. Select a black swatch, click **OK**, and click **Apply**. A dialogue prompt will ask if you want to change the **Foreground** text and **Mouse Drag Lines** to white.

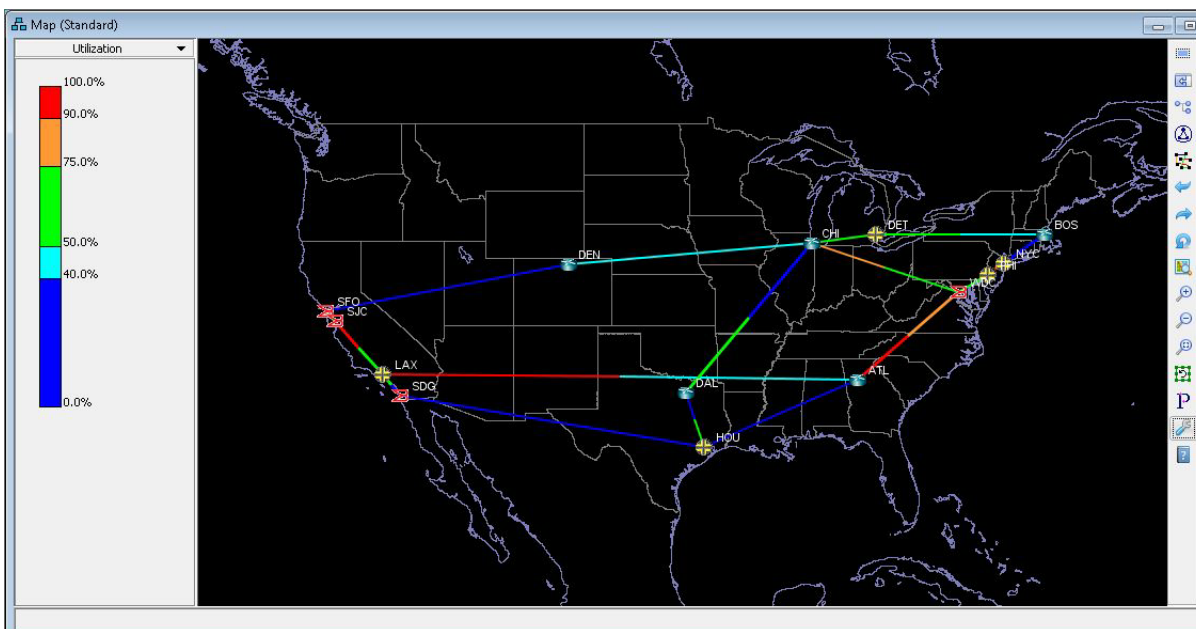


Figure 22: Black Background Map

Topology Layout

Right-click on the Map to access the **Layout** sub-menu. The layout descriptions are below.

Distribute Selected Points

Select the nodes that you would like to distribute. This will attempt to evenly spread out the nodes from each other. Once you click on **Distribute Selected Points**, a cross-hair cursor appears for you to set the network epicenter and outer distance. The first click will be the epicenter of the network, and the second click will be the radius distance to distribute the elements.

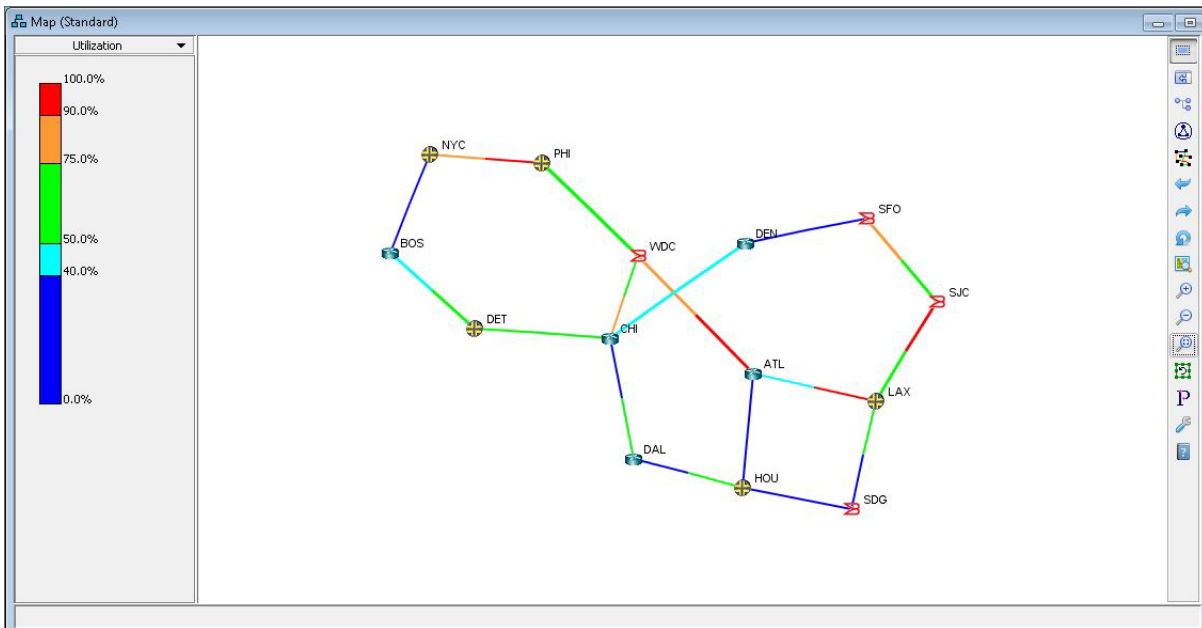


Figure 23: Distributed Layout

Circle Nodes

Select the nodes that you would like to form a circle. Once you click on **Circle Selected Nodes**, a cross-hair cursor appears for you to set the network epicenter and outer distance. The first click will be the epicenter of the network, and the second click will be the radius distance to distribute the elements.

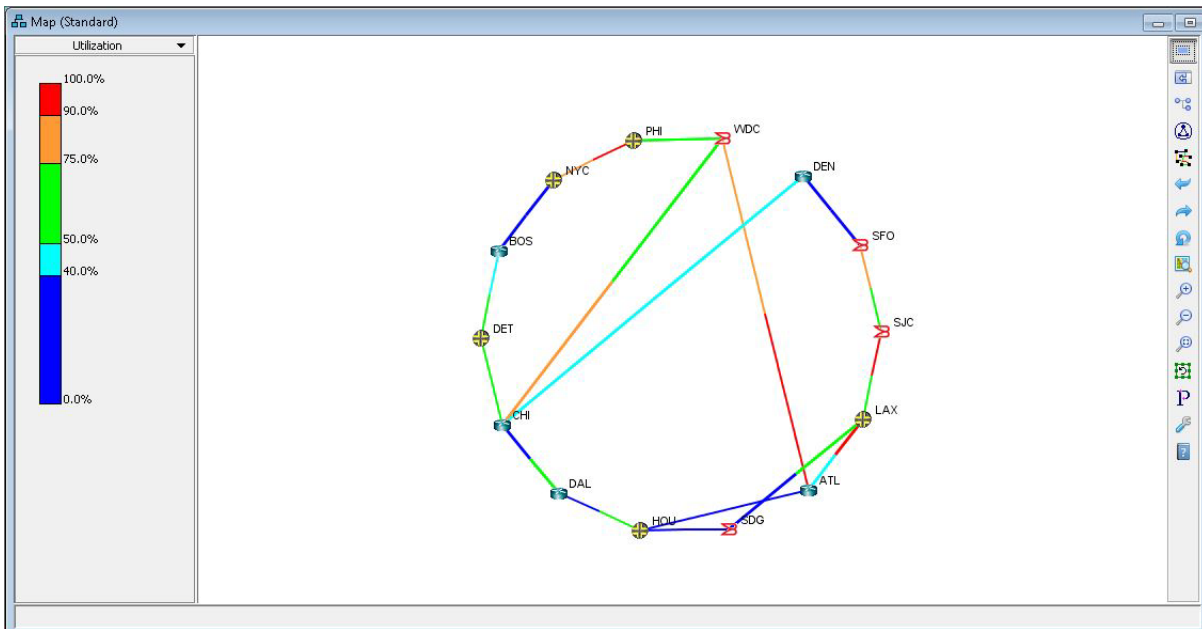


Figure 24: Circle Layout

Straighten Nodes

Select the nodes that you would like to place on a horizontal line. Once you click **Straighten Selected Nodes**, the nodes will be straightened.

Reset by Lat/Lon

Reset by Lat/Lon places the nodes based on their geographical coordinates. The latitude and longitude information must be in the muxloc file.

Cut, Copy, and Paste

You can perform quick modifications to the Map by utilizing the cut, copy, and paste functions. If the node contains router-config XML data, it is carried over as well.

Select the nodes you wish to cut or copy. Any links that (both) originate and terminate at the selected nodes will be included as well. Right-click on the Map and select **Cut Selection** or **Copy Selection**.

Right-click on the Map where you wish to paste the nodes and select **Paste**. A dialogue box will ask if you wish to create a group for the pasted elements.

Every pasted element will be appended with a number to uniquely identify it.

Zooming In and Out and Using the Navigator Window

Toggle the Navigator icon to open or close the Navigator window in the bottom right pane. The Navigator pane gives a view of the entire topology with a black rectangle around the area in view in the main Map window. Drag the black rectangle around to pan across the Map.

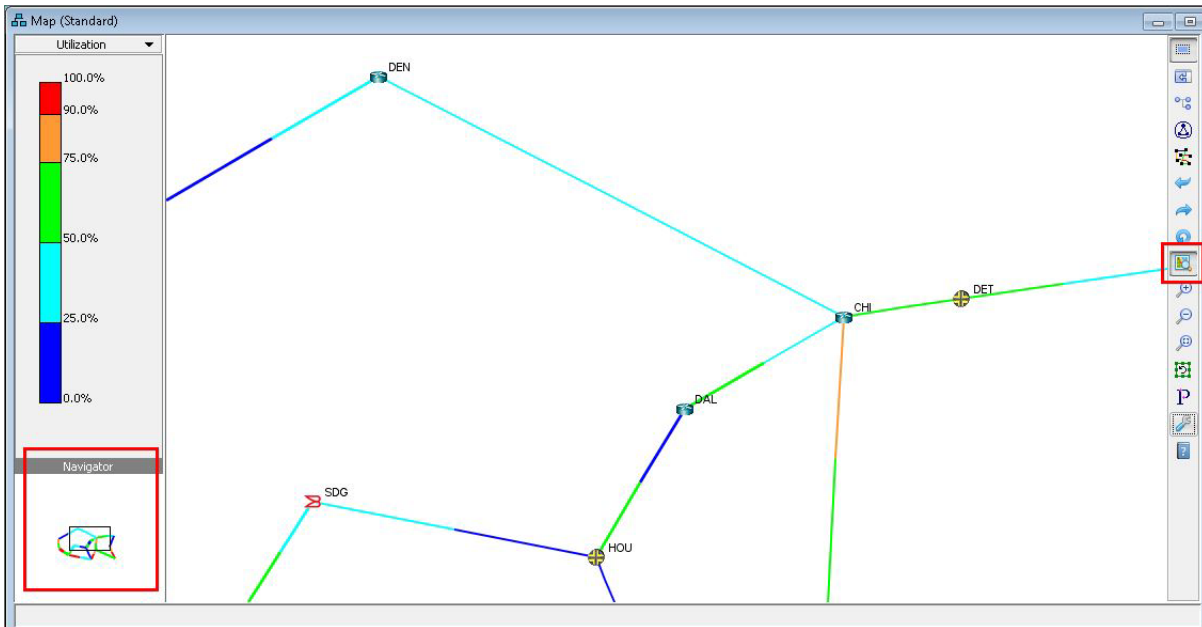


Figure 25: Navigator Window

Click the **Zoom In** icon. Then hold down the mouse button and drag to form a target for the zoom in area. The mouse wheel up also zooms in by fixed increments.

Click the **Zoom Out** icon. Each click zooms out by fixed increments. The mouse wheel down also zooms out by fixed increments.

Click the **Zoom To Fit** button. This will auto zoom to fit all elements into the Map.

The Map right-click menu has additional Zoom options.

The **Zoom To Fit Selection** will auto zoom to fit all selected elements into the Map.

The **Zoom To Fit Region** allows you to zoom to a particular geographical region.

Grouping Functions

Right-click on the Map to access the **Grouping** feature. This allows you to group or ungroup nodes.

The Map toolbar includes shortcuts for the **Group Selected** and **Ungroup Selected** actions.



Informational Note: You can create groups within groups. However, if you try to group two items that do not originally share a common group, they will be taken out to form their own group.

Expanding and collapsing a group are very different from grouping and ungrouping. Grouping establishes which nodes are in which groups and ungrouping is used to undo a grouping. Meanwhile, expanding a group does not change the group arrangement but displays what is inside a group. Similarly, collapsing a group does not change the group arrangement but hides what is inside a group.

You can view the groups and associated nodes in the left pane of the Map by using the **Node List** feature.

AutoGroup enables you to group by special properties. A dialog box allows you to choose different levels of grouping. Click **Done** if you have no further grouping level to request.

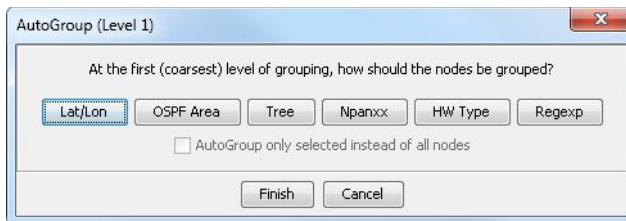


Figure 26: AutoGroup Dialog Box

The first level will be used to group all the points. The second level will be used to create subgroups within each group. The third level will be used to create subgroups within each subgroup. And so forth...



Informational Note: Creating groups using **AutoGroup** will overwrite any existing groups.

Viewing Your Network Model By Subview

In the drop-down menu of the left pane of the Map window, select **Subview > Protocols**. The list of Protocols may vary depending upon the protocols available in the current network.

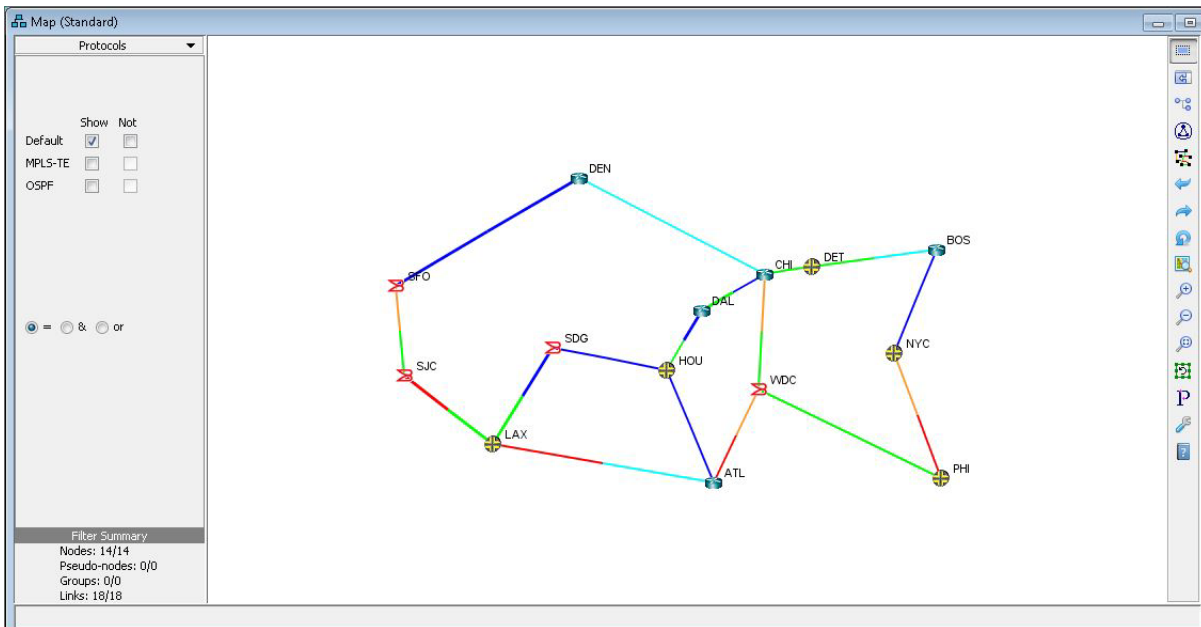


Figure 27: Protocols Pane

The **Default** subview is chosen by default.

Choose a different protocol subview (for example, **OSPF**).

You can also use the “&” and “or” functions. “=” is selected by default. To view different logical combinations of protocols. Select the two protocols you want to view together, such as MPLS and OSPF, and the union of the nodes supporting those will be displayed. Using “&” instead of “or” shows the intersection.

To hide nodes that are not connected to any other nodes in the network, you can select **Filters > Advanced** from the map drop-down and then check the checkbox next to **Hide Isolated Points**. There is also a **Hide Isolated Points in Subview** checkbox. This option hides nodes that appear disconnected from other nodes in the *current* subview.

Map Views

Multiple map views can be saved for a network, which includes the grouping, graphical coordinate information, zoom, filter, and legend settings. Right-click on the Map to access the **Map Views** feature.

To save a map view, click **Save Map** and provide a name and description.

The options Local, Private, and Shared will set access permissions for these map views. Local saves the map view to the spec directory and is accessible to anyone who can open the specification file. Private and Shared are intended specifically for Live Networks. Private saves the map view to your local Live Network directory and is accessible to the owner. Shared saves the map view to the public Live Network directory and is accessible to anyone who can view the Live Network.

To load a map view, select the entry from the list and click the **Load Map** button. Use the directory drop-down to select between loading from the Local or Live Network directories.

To update an existing map view, select the entry to update, click **Save Map**, and use the same name. This will overwrite the previous map view.

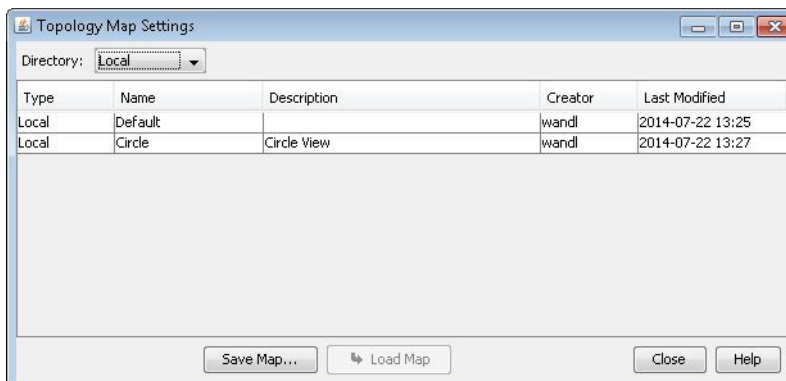


Figure 28: Map Views

Copy Map Layout

If you have more than one Map open such as the Standard and BGP map, you may copy the layout from one map to another. To copy the layout of one map and copy it to another map, you may select from the **Network > Maps > Copy Map Layout** menu.

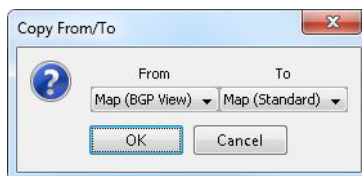


Figure 29: Copy From/To Window

In the **Copy From/To** window, the user may select which map to copy the layout from, and which map to apply the layout to. In the example above, we will apply the BGP map's layout to the standard map. This is ideal if you make a lot of node rearrangements to one map and want the same rearrangements for another map.

Click **OK** when the appropriate setting has been selected.

Export... and Print...

Print...

Right-click in the Map and select the **Print...** option to print the Map window. A dialog box will ask to print the whole topology window or the map only.

Select **Whole map window** to save the whole map window including title bar, legend, and toolbar. Select **Map only** to save only the map portion of the window.

Export...

Right-click in the Map and select the **Export...** option to save the Map window to a file

Select the folder in your client machine from the **Save In** drop-down menu that you want to save your file to, or click on the Create New Folder icon to create a folder.

Select the file type from the **Files of Type** drop-down box and type the name of the file you want to save to where it says **File Name**.

Different options will be enabled in the **Show in Output** section depending on the file type..



Informational Note: Both the "Print..." and "Export..." options can produce postscript files. The latter option produces Adobe PostScript Level 2 which may not be supported by old versions of postscript readers. For the former, the PostScript Level may be dependent upon the printer selection.

Export to Visio

This section explains how to export a graphic of the topology display to Visio where it can be manipulated and altered. The following formats are supported:

- VDX format for Visio 2003,
- CSV format for Visio 2002 and earlier. After opening the CSV file in Visio 2002, the resulting Visio diagram can then be saved to a Visio .VSD file using Visio's **File > Save As** dialog.

Export Map to Visio

Adjust the map settings (node positions, labels, colors, and so on.) so that what you export is currently in view.



Informational Note: *Only the portion of the network that is in view* will be exported. The boundaries of the drawing (up to the margins) coincide with the boundaries on the screen when you exported. If you wish to export the whole network, use the **Zoom to Fit** icon (square within a magnifying glass) on the upper right icon toolbar of the topology window before you export.

Right-click on the map, select **Export...**

Select a directory on your client machine in which to save the Visio file. Then select the Visio file type: Visio 2003 and later (VDX), or Visio 2002 and Earlier (CSV).

If you wish to display a title or comment, specify it under the **Titles, etc.** section and make sure the corresponding item is checked in the **Show in Output** section. You may also choose to display the legends and/or Date/Time.

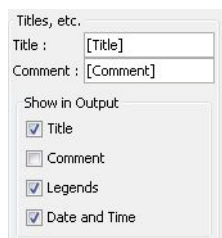


Figure 30: Export to Visio Options

Click **Save** to continue. The system then displays a dialog box with page and margin sizes. Landscape is recommended (see “Visio Tips” for tips on the aspect ratio). If you want to print or plot onto large paper sizes (such as 60 by 36 inches), change the width and height (in inches) here. The Country Map Resolution can be changed to coarsen the map resolution in order to reduce the Visio output file size. The default resolution is approximately 1 pixel (1/77th of an inch). Increasing this number will make the country map coarser by increasing the length of the line segments making up the map.

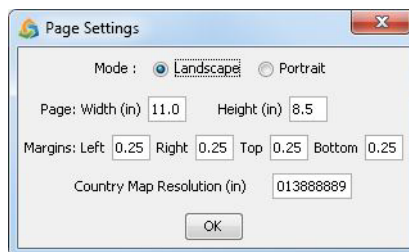


Figure 31: Page Settings Window

Click OK to see the message “Export completed successfully.”

Setting Up the Stencils (for Visio 2002 and earlier only)

If you saved to the VDX format (Visio 2003), skip to the section <Link>Opening the Visio Drawing on 7-50. For CSV format, please follow the steps below to ensure that the correct stencils are being used.

In order to get the correct symbols for nodes, links, groups and text fields, you need to provide Visio with IP/MPLSView’s stencils. The following instructions explain how to do this.

Find the Visio files `WandlStencil.vss` and `WandlTemplate.vst` in your client machine’s installation directory and copy them into Visio’s Solutions directory.

- The files `WandlStencil.vss` and `WandlTemplate.vst` are in the client machine's installation directory, which defaults to `C:\Program Files\wandl\NPAT4.3` or `C:\Program Files\wandl\IP-MPLSView4.3` for Microsoft Windows clients and to `/u/wandl/client/visio` for Solaris clients.
- For a standard Visio installation, Visio's Solutions directory will be in `C:\Program Files\Visio\Solutions`. In Visio 2002, the Solutions directory will be in `C:\Program Files\Microsoft Office\Visio10\1033\Solutions`. (the 10 and 1033 may differ on your machine)

Opening the Visio Drawing

Start Visio and press **Cancel** to get out of any welcome screens, if displayed ("Choose a Drawing Template" in versions 4.5/5.0 or "Welcome to Microsoft Visio" in Visio 2000).

Open the saved file from the **File > Open** menu option.

(VDX format) Select the **"Enable Macros"** button when asked whether to enable macros. If you are given a dialog about XML warnings, click "OK" to continue.

(CSV format) A Visio File Converter window pops up. Accept the defaults it offers by pressing **OK**. If you are given a dialog box asking if you wish to enable macros or not, select the **"Enable Macros"** button. You should see the map being loaded into Visio.

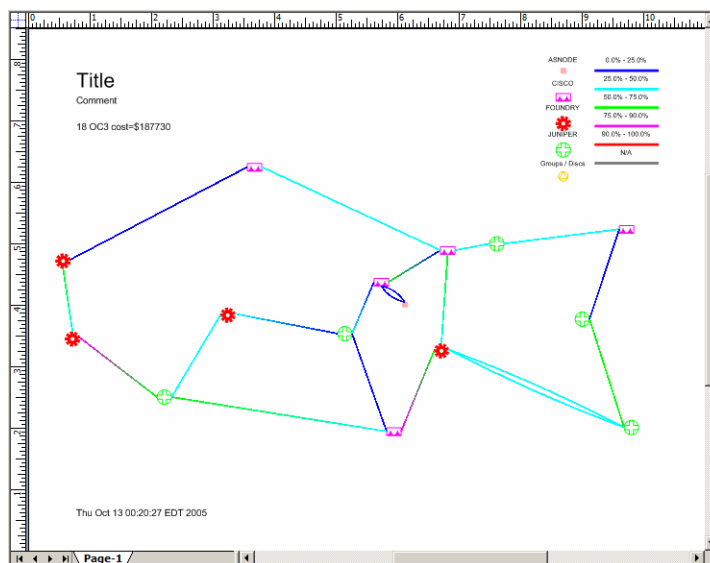


Figure 32: Example Visio Output

Page Setup

Depending upon the page setup mode you chose (landscape or portrait), you may need to change the Visio settings from portrait to landscape or vice versa.

Version	Instructions to change setup mode to Landscape
Visio 4.5	Go to Edit > Drawing Page > Size & Scale, and choose Landscape.
Visio 5.0	Make changes under File > Page Setup. Then press the button Print Setup that this window contains and make further changes.
Visio 2000	Go to File > Page Setup, and choose Landscape in both the "Page Size" tab and the "Print Setup" tab.
Visio 2002	No changes necessary.
Visio 2003	No changes necessary.

Setting Link Colors and Curvature

For the VDX format, the link colors and curvature should be automatic. However, for the CSV file format, the links in the map will be displayed as black straight lines. To give them the correct colors and curvature, go to Visio's **Tools** menu and choose the following items in succession: **Macros > IP/MPLSView > LinkColorsAndCurvature**. See the last image in Figure 33 for an example.



Informational Note: For some versions, it may be necessary to set up Visio to run macros with Visual Basic for Applications (VBA). If the version of Visio has not been set up properly, "IP/MPLSView" will not appear in the Macros menu.

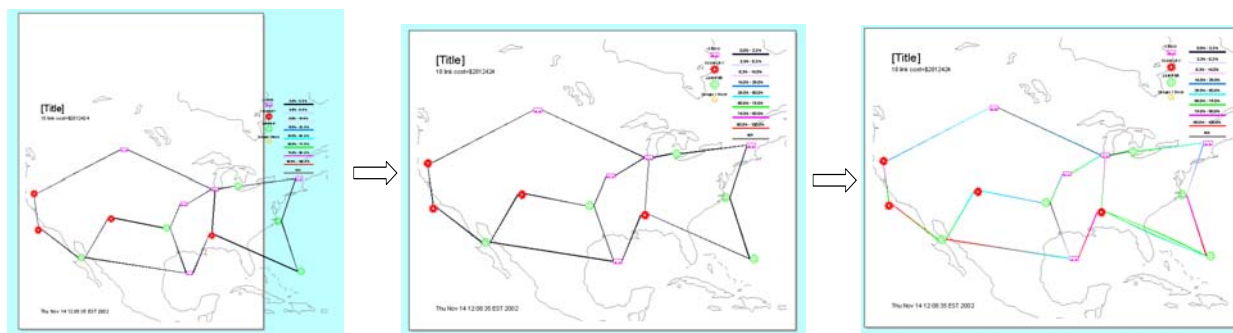


Figure 33: Portrait to Landscape Conversion Followed by Setting the Link Colors and Curvatures

Handling Groups

If the export includes expanded groups, the discs will appear in Visio. Sometimes the discs lie on top of nodes and links, hiding them. And if the discs are all the same color and there are groups within groups, it may be difficult to see the lower level discs. To fix this, you may want to adjust the coloring of large groups and/or send them to the back.

In order to send multiple discs back all at one time, you can select multiple discs before performing the desired option.

To select one or more discs, click on the first one; then shift-click on the others (they will be outlined in light blue). Then press F3 to fill or CTRL-b (Visio 2000 and earlier) or CTRL-SHIFT-b (Visio 2002) to send to back.

While the network elements may fit entirely within the Landscape view, the groups sometimes do not. Because of the aspect ratio, the discs will generally become ellipses. Hence you may also want to resize a disc by selecting a disc and then dragging the green handles.

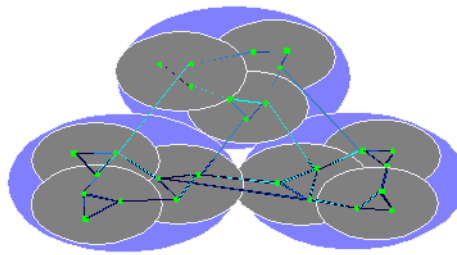


Figure 34: Visio Drawing After Coloring and Sending Discs to Back

Topic	Instructions	Shortcut Keys
Fill selected disc(s)	Right-click one of the selected disc(s) and select Format>Fill... . Then select a color.	F3
Send selected shape(s) to Back	Right-click one of the selected disc(s) and select Send to Back .	CTRL-SHIFT-b (Visio 2002) CTRL-b (Visio 2000 and earlier)
Send all discs to the back	Tools>Macros>IP/MPLSView>GroupDiscsToBack	
Resize a disc	Select a disc. Then drag the green handles.	

Visio Tips

Subject	Tips
Dragging a Node	<ul style="list-style-type: none"> Make sure you are clicking on the node and not the link or any displayed map borders. It helps to zoom in around the node so you can see it clearly. Furthermore, clicking inside the node and not on the edge of the node may help to avoid accidentally resizing the node. If you drag a node, the links connected to it will move with it. The opposite endpoint of the link will remain fixed. But if you drag a link, it will be torn away from its nodes. This is the correct behavior for Visio's connectors.
Changing a Link's Curvature	Click on the link, then drag the link's center point to make it curved.
Aspect Ratio	The aspect ratio (width/height) should be about the same in the Visio drawing as in the map, or the image will be distorted. For instance, do not export in portrait mode when the map is wider than it is tall; resize the map first by dragging its borders.
Editing the Stencil	(CSV format) Make global changes to the drawing by editing the stencil WandlStencil.vss. For instance, to turn all the diamonds into discs, just edit the master diamond in the stencil, then re-open the .csv file. Or replace the master with one of the many symbols in Visio's Network Diagram stencils.
Editing the .csv file	(CSV format) Consider editing the .csv file if you want to change a node's hardware type. The format of the .csv files is specified in Visio's online help files (search on importing text and text import record types). As always, use caution when editing a computer-generated file.
Handling Dotted Lines	The Export to Visio feature supports dotted line styles, as long as the dotted line is limited to one color. Solid lines can be exported in up to two colors (for example to show the utilization in both directions on a link). However, it is not possible to export a link in two colors to show the utilization, and have it be dotted at the same time.

Scalable Vector Graphics

Scalable Vector Graphics (SVG) is an open standards XML-based vector graphics language. SVG files can be viewed from within a web browser such as Firefox, Internet Explorer (version 9 or later), Opera, or Safari. Alternatively, you can use a third party SVG plugins such as Adobe SVG Viewer. There are also tools to import, export, and manipulate SVG graphics. For example, Adobe Illustrator and CorelDraw are third party tools can import and export the SVG format.

To save to SVG format on your client machine, select the **Export...** option from the map's right-click menu and save using the **Scalable Vector Graphics (SVG)** format. Additionally, you can choose to include a title, comment, legends, and date/time in the SVG file.

Note that the resulting SVG file depends upon existing settings in the topology map. These settings include whether the world map should be displayed, the boundaries of the world map, and whether node and link labels should be displayed.

Chapter 8

Viewing and Saving Network Information

This chapter explains how to view and save network information, such as node, link, and demand information. Detailed information on nodes, links, and demands may be retrieved directly from the map through right-click popup menus or by selecting the network element from the **Network** menu. Network information can also be seen from the status bar, console, and file manager.

When to use

Use these procedures to retrieve node, link, and demand information and find out how additional demands will be routed. Certain features are device-specific therefore the GUI may vary according to your network's hardware type and your license file.

Prerequisites

You should have started up the IP/MPLSView software and opened a specification file as described in Chapter 4, Opening and Closing a Network Project. You should also have a basic overview of the topology map features as described in Chapter 7, Using the Topology Window.

Related Documentation

How to view site and owner information is explained in the Chapter 11, Modifying Sites and Owners. For information on how to view custom rate information, see Chapter 13, Modifying Link Pricing. Although it is a chapter on modification, the interface is very similar. Some of the other features not mentioned may be device-specific modules.

See the separate document relevant to the modules you have. You may also want to refer to the IP/MPLSView Java-Based Graphical User Interface Reference network chapter.

Recommended Instructions

Following is a high-level, sequential outline of the process of viewing network information and the associated, recommended detailed procedures.

This tutorial uses the sample network specification file `/u/wandl/sample/IP/fish/spec.mpls-fish`. If you do not have the required license to use this example, you may edit the network project, as described in Following Along With the Examples in This Manual on page 1.

Note that using a different hardware model may affect the routing and therefore the results you get may be different than that obtained for the router hardware used in this example.

1. View basic network information.
2. Open the **View Nodes** window and specify search criteria to filter nodes that meet a particular criteria as described in step 1 on page 57.
3. View basic node and link information via the map popup menus, as described in step on page 59.
4. Save a report of the link information as described in step 1 on page 61 and view the report from the **File Manager** as described in step 4 on page 62.
5. Access demand information for the link using a button in the link window as described in step 1 on page 62.
6. Try various options in the demand window, including filtration of demands going through the link and the “**Show Path**”, “**BW+Ovhd**”, “**Details**”, and “**Show Traffic**” options as described in step 1 on page 63. Save **Console** messages as described in step 2 on page 63.
7. Check if sufficient bandwidth exists in the network to place additional user demands between two nodes and view the path between two nodes as described in step 1 on page 64. Also survey the other **Capacity** options as described in step 5 on page 65.
8. View aggregated demand information as described in step 1 on page 66.

Detailed Procedures

1. Click on the **Network** menu from the main window menu bar to view the different types of network information. These options may vary depending upon the hardware of the current network and your license file.

Not all of these menu options will be described in this chapter. Please consult the following user manuals for device-specific features.

Manual	Network Menu Items
Voice	Voice
ATM/PNNI training guide	Card Cost, PNNI
Transport-Specific training guide	Ring
Router Feature Guide for IP/MPLS	Aggreg Tunnels, BGP, Class of Service, Hardware Equipment Cost, Interfaces, IP VPN, LSP Tunnels, Multicast Group, OSPF Areas, Static Route Table
IP/MPLSView Java-Based Graphical User Interface Reference and/or this manual	Admin Weight, Aggreg Demands, Custom Rates, Demands, Domains, SRLG/Facilities, Hardware Equipment Cost, Links, Link Rules, Nodes, Owners, Path & Capacity, Pricer, Service Type, Service Profile, Show Site Demands, Sites, Summary, Templates, User Parameters

2. Select **Network > Summary** to view the network summary window.

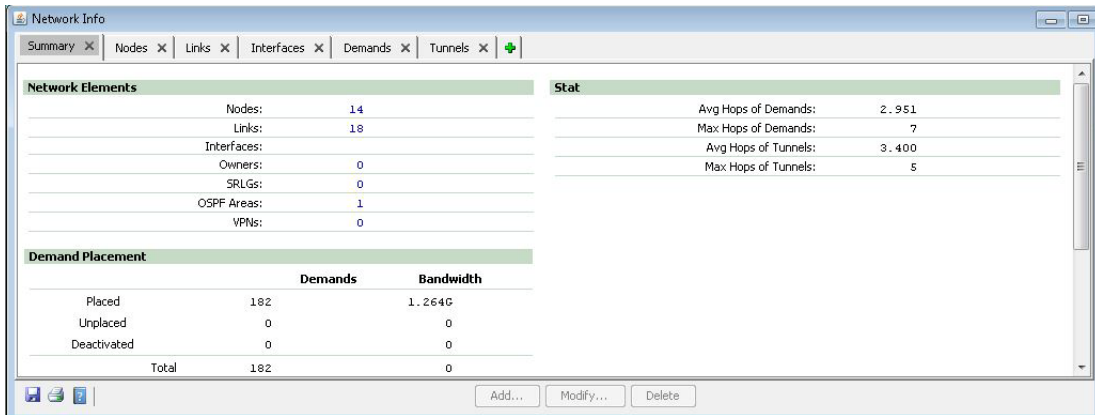


Figure 35: Network Summary

3. This window provides you with a high level understanding of your network including the hardware type, cost, number of nodes and links, average and maximum hops of demands, and a breakdown in total number of demands and total demand bandwidth according to whether a demand is placed, unplaced, or deactivated.

Node, Link, and Demand Searches

1. To retrieve more detailed network information for nodes, links, and demands, select the Nodes tab. Or use the **Network > Elements** menu items for **Nodes**, **Links**, and **Demands**.

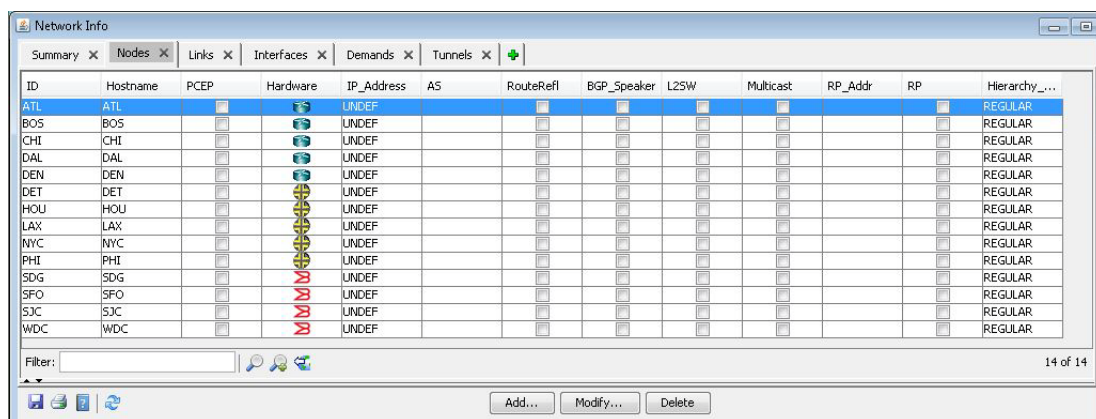


Figure 36: Nodes tab

2. Now try selecting the **Links** tab (same as **Network > Elements > Links**).

- For the node and link windows, there are more attributes than those displayed in the table. To customize the columns included in the table, right-click on the column header and select **Table Options**. Select the columns you want to view from the **Available Item(s)** list, add them to the **Selected Item(s)** list, and rearrange the column order using the up and down arrows. Press **OK** for the changes to be applied.

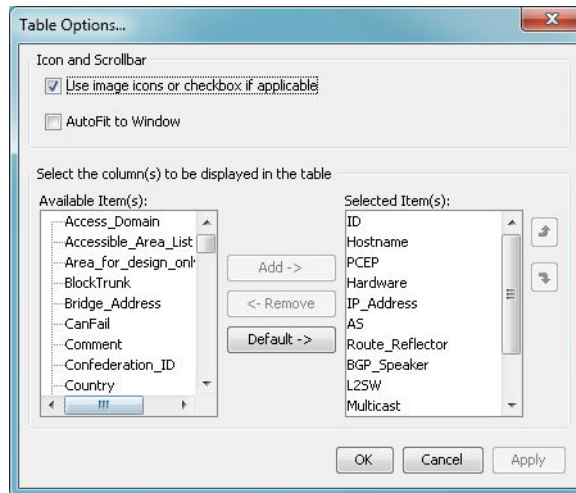


Figure 37: Table Options

- The columns in the table can be sorted by clicking on the column header or right-clicking on the table header and selecting **Sort Ascending** or **Sort Descending**.
- To filter through the list of nodes, links, or demands with a certain search criteria, select the node view and click the **Search by Property** button.

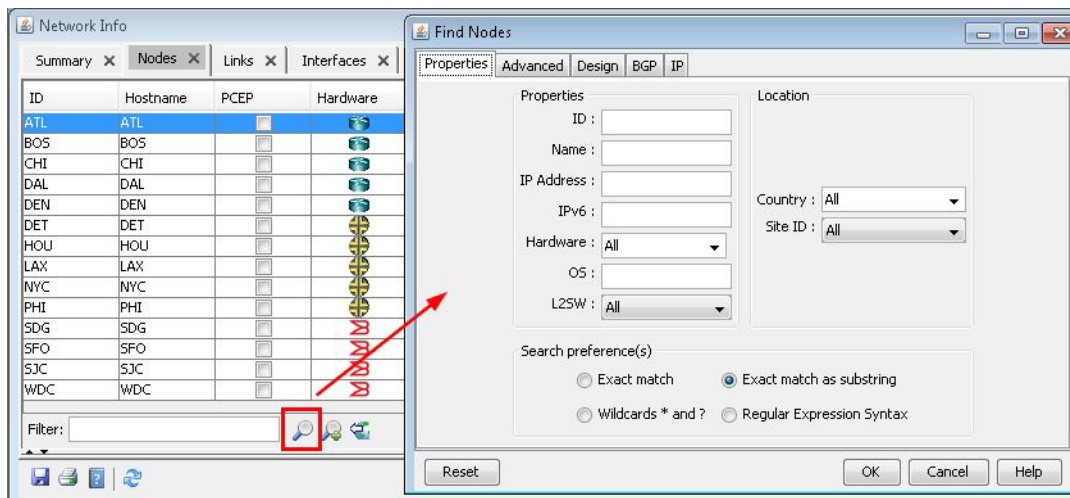


Figure 38: Find Nodes Window

6. In this window, you may enter a search criteria based on a wide range of parameters in the available tabs. Select the **Hardware** drop-down menu and specify the hardware type "JUNIPER" (or an appropriate hardware type for your network) and click **OK**. The nodes that match this hardware type will be fetched and listed in the upper table of the **Nodes** window. Close this window.



Informational Note: Available search preferences are exact match, exact match as substring, wildcards, and regular expression syntax. See Appendix B of the IP/MPLSView Java-Based Graphical User Interface Reference for more details.

Advanced Search

You can follow the same procedure with the **Advanced Search** button for **Links** and **Demands**. For link and demand queries, you can use the ">", "=", and "<" operators in your search. For example, in the link query **Utilization** tab where it says **Util(A->Z)**, select ">" and then type 0.7. This query will return links whose utilization in the A to Z direction is greater than 70%. Similarly, for the demand query, you can search for demands with bandwidth requirements greater than, equal, or less than a certain amount.

You can also enter your own query string using 'and' and 'or' logical operations. The Keys, Relations, and Current Values sections of the window give you possible fields that can be checked, the relation (=,!=,<,>,<=,>=) to check, and the value to compare the key to. For example, for a text filter for the link window you could specify, `Avail_BW_ZA < 1M or Avail_BW_AZ < 1M or Util_AZ > 0.9 or Util_ZA > 0.9`, to find all links with available bandwidth less than 1M or utilization greater than 0.9 in at least one direction. This can be directly typed into the query field. The **Insert** buttons provide shortcuts to typing.

Node, Link, and Demand Map Menus

Node, link and demand properties can also be retrieved directly from the map by right-clicking over a link or a node. To learn more about the use of the other right-click menu items related to the topology window, refer to Chapter 7, Using the Topology Window.

1. To get basic node information, place your mouse pointer over a node. Notice the status bar at the bottom of the map will display the node name, hardware type, and location information. The content of the message in the bottom bar of the map window can also be modified as described in Bottom Bar Labels on page 39.

2. To get basic link information, place the pointer over a link. Information such as link, trunk type, and planned utilization is displayed in the status area. If the planned utilization for the A->Z direction differs from the Z->A direction, the link will be colored in two halves, and the planned utilization will be displayed in the format “(planned utilization for the A->Z direction, planned utilization for the Z->A direction)”.

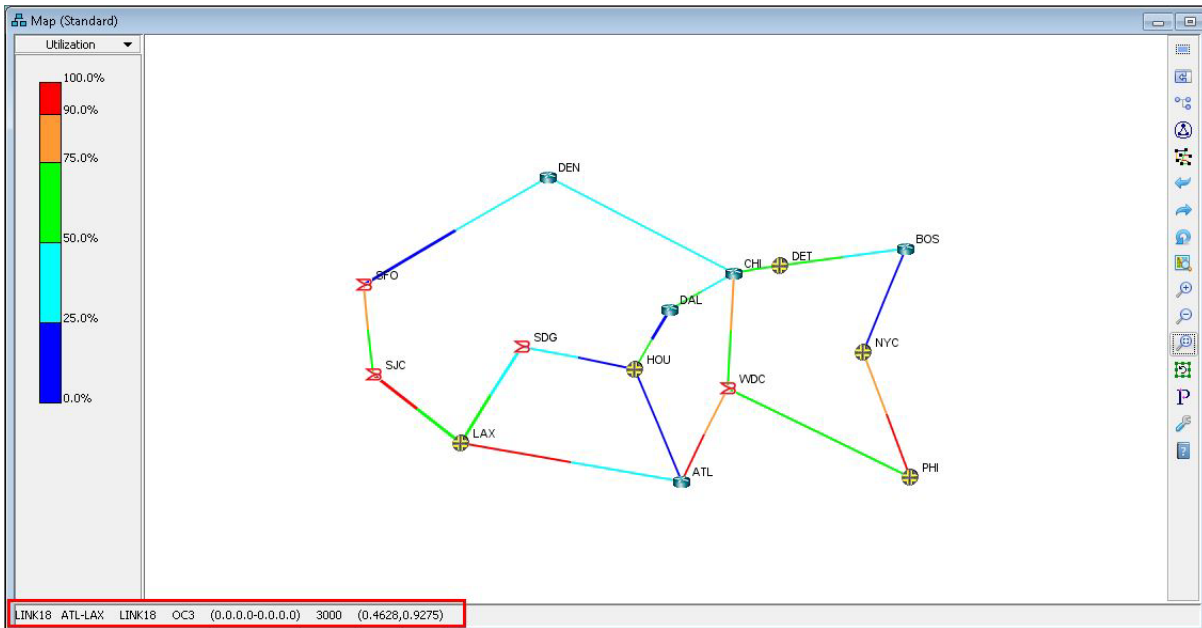


Figure 39: Bottom Bar Displays Link Information

3. Node and link properties can also be displayed on the map by using the map right-click menu's **Labels** options and customizing the label.
4. You can view the node information from the Map by right-clicking over a node and selecting **View > Nodes under Pointer**. If there are multiple nodes overlapping, this will display the information for all those nodes.
5. You can view information for multiple selected nodes by <Ctrl>-clicking the nodes and then right-click one of them and select **View > Selected Nodes**.
6. Toggle the up and down arrows to see more details and access the informational tabs.

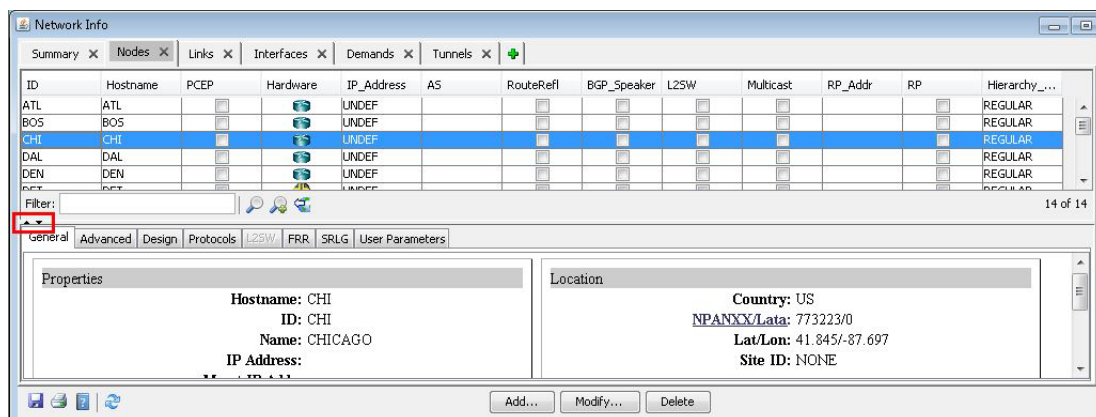
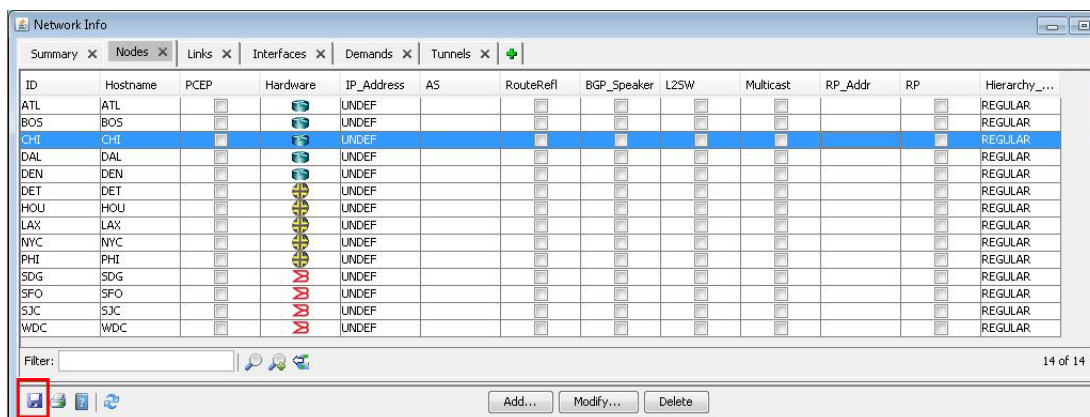


Figure 40: Detailed Information

- Clicking on entries in the **Nodes** tab will highlight the selected node on the map.
- You can right-click over a link and select **View > Links under Pointer** from the right-click menu to get a **Links** window. Similar to the display of node information, the information for each link is separated into tabs where each tab presents different properties regarding the selected link.
- You can view information for multiple selected links by <Ctrl>-clicking the links and then right-click one of them and select **View > Selected Links**.

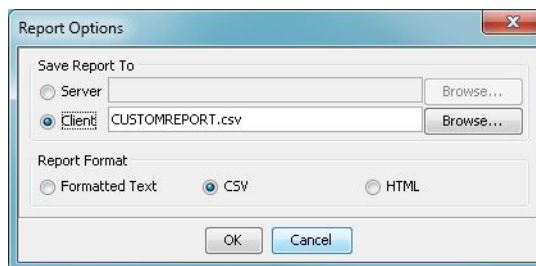
Saving Network Info to a Report

- To save the data in the currently open **Network Info** window, click the **Save** button.
- You can first customize the columns to display and save by right-clicking the column header and selecting **Table Options...**

**Figure 41: Saving Network Info**

- You can save the report to the IP/MPLSView server or your client machine. Click **Browse** to set the saved directory.

Available formats to save the report are Text, CSV, and HTML.

**Figure 42: Save Report**

Informational Note: If the file already exists on the server in the chosen path you are trying to save your file to, you will be asked whether or not to overwrite it or cancel the save operation.

- To view a report saved on the server, open the **File Manager** and navigate to the directory containing the output file. Then double-click on the report file. Click on the “X” box at the top right corner to close this window. Note that the “**Print**” function will print on your client machine and the “**Save**” function will save to your server machine.

Viewing Demands

- In the **Links** tab, right-click an entry and select **Demands on/thru Link**. You can also view demands through the link by right-clicking the link on the Map and selecting **View > Demands on/thru Link**. If no demands are found, you may get an empty result.

ID	NodeA.ID	IP_A	NodeZ.ID	IP_Z	BW	Type	Pri	Pre	Current_R...	Configured	Secondary	Owner
flow2	ATL		CHI		730.017K	R	02	02	ATL[(T=RAT...	No Pref.		NONE
flow3	ATL		DAL		418.017K	R	02	02	ATL--HOU--...	No Pref.		NONE
flow4	ATL		DEN		520.016K	R	02	02	ATL[(T=RAT...	No Pref.		NONE
flow6	ATL		HOU		418.017K	R	02	02	ATL--HOU--...	No Pref.		NONE
flow18	BOS		HOU		558.028K	R	02	02	BOS[(T=RB...	No Pref.		NONE
xflow3	DAL		ATL		441.231K	R	02	02	DAL--HOU--...	No Pref.		NONE
xflow6	HOU		ATL		441.231K	R	02	02	HOU--ATL--...	No Pref.		NONE
xflow41	NYC		DAL		4.801M	R	02	02	NYC--PHI--...	No Pref.		NONE
xflow42	PHI		DAL		1.216M	R	02	02	PHI--WDC--...	No Pref.		NONE
xflow46	WDC		DAL		4.689M	R	02	02	WDC--ATL--...	No Pref.		NONE
xflow65	NYC		HOU		4.801M	R	02	02	NYC--PHI--...	No Pref.		NONE
xflow66	PHI		HOU		1.216M	R	02	02	PHI--WDC--...	No Pref.		NONE
xflow70	WDC		HOU		4.689M	R	02	02	WDC--ATL--...	No Pref.		NONE

Demands through link: ATL-HOU 13 of 13

Figure 43: Demands Through the Link Window

- The **Demands** tab table allows a maximum of 1,000,000 demand entries to be displayed continuously. Use the paging option to set the number of demands to view in a single page, and arrows to navigate through these pages.

ID	NodeA.ID	IP_A	NodeZ.ID	IP_Z	BW	Type	Pri	Pre	Current_R...	Configured	Secondary	Owner
flow1	ATL		BOS		730.017K	R	02	02	ATL--WDC[...	No Pref.		NONE
flow2	ATL		CHI		730.017K	R	02	02	ATL[(T=RAT...	No Pref.		NONE
flow3	ATL		DAL		418.017K	R	02	02	ATL--HOU--...	No Pref.		NONE
flow4	ATL		DEN		520.016K	R	02	02	ATL[(T=RAT...	No Pref.		NONE
flow5	ATL		DET		418.017K	R	02	02	ATL--WDC[...	No Pref.		NONE
flow6	ATL		HOU		418.017K	R	02	02	ATL--HOU--...	No Pref.		NONE
flow7	ATL		LAX		520.016K	R	02	02	ATL--LAX--...	No Pref.		NONE
flow8	ATL		NYC		20.490M	R	02	02	ATL--WDC--...	No Pref.		NONE
flow9	ATL		PHI		520.016K	R	02	02	ATL--WDC--...	No Pref.		NONE
flow10	ATL		SDG		418.017K	R	02	02	ATL--LAX--SDG	No Pref.		NONE
flow11	ATL		SFO		520.016K	R	02	02	ATL--LAX--S...	No Pref.		NONE
flow12	ATL		SJC		730.017K	R	02	02	ATL--LAX--SJC	No Pref.		NONE
flow13	ATL		WDC		27.978M	R	02	02	ATL--WDC--...	No Pref.		NONE
flow14	BOS		CHI		870.028K	R	02	02	BOS--DET--...	No Pref.		NONE
flow15	BOS		DAL		558.028K	R	02	02	BOS--DET--...	No Pref.		NONE

100 of 182 (page 1/2)

Figure 44: Demands Page Navigation

3. Right-click a demand and select **"Show Path"** to highlight the demand's current path on the map.

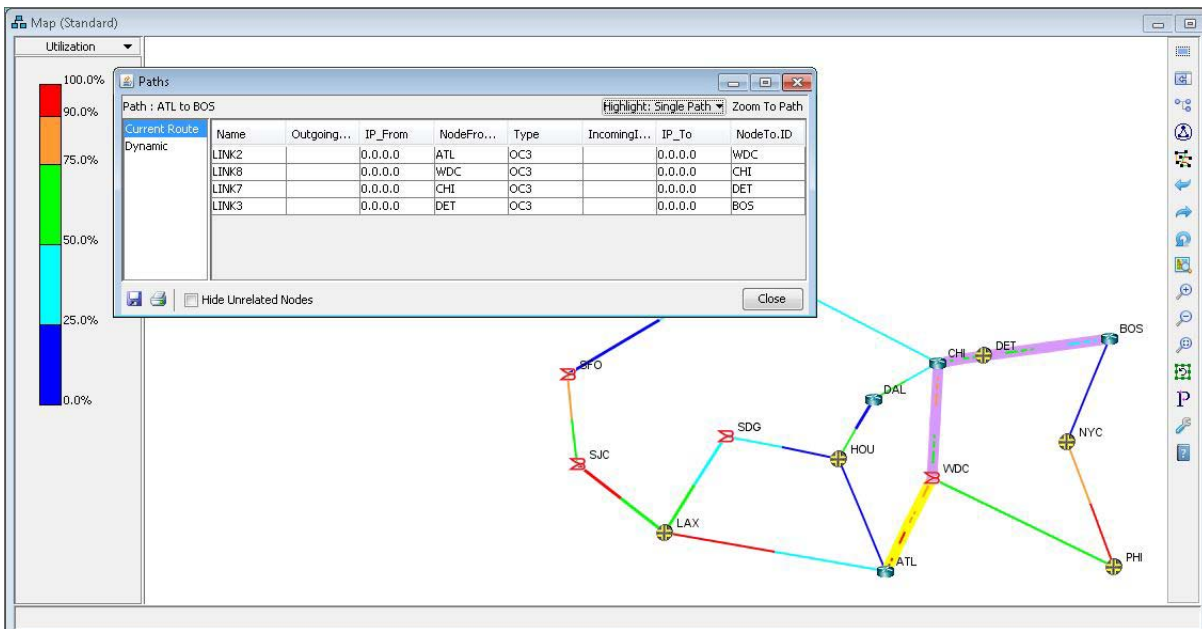


Figure 45: Demand Show Path

If path animation is turned on, you will be able to see the direction of the path by the moving line segments.

If you have an MPLS-enabled network, a purple highlight may appear along the path to indicate the passage through a tunnel. If you look at the **Current Route** column in the **Demands** tab, you may see tunnel paths in brackets.

4. Now select **"Highlight All"** in the **Demands** tab. "Highlight All" differs from the other path and route displays, as this feature displays just a line segment between the source-destination pairs of all the demands listed in the table.

Obtaining and Saving Network Information in the Console

1. In the **Demands** tab, right-click a row and select **"BW_Ovhd."** The console will be populated with bandwidth and overhead information.
The **Console** window provides an additional area where network details can be displayed. Additionally, it provides an alternate method of input for the user. For some operations, an **Input** text field will appear within the console window. Before attempting to enter input, the user should ensure that the Console is the current active window (by clicking inside the **Input** text field).
2. To record the **Console** information to a file on your client machine as it is generated, right-click on the console box and select **Trace** from the right-click menu if the checkbox next to **Trace** is unchecked. Then, specify a filename on the client machine in which the console trace should be saved. If the file already exists, you will be asked whether or not to overwrite it, append data to it, or cancel the save operation.
3. To end the trace, right-click within the **console** window and select **Trace** again to uncheck the box.

Alternatively, if you are using a PC client, you can save console text by selecting the text with your mouse (or using <Ctrl>-a to select all text in the **Console** window) and then pressing <Ctrl>-c to copy the text. Then you can use <Ctrl>-v to paste to another window on your PC.

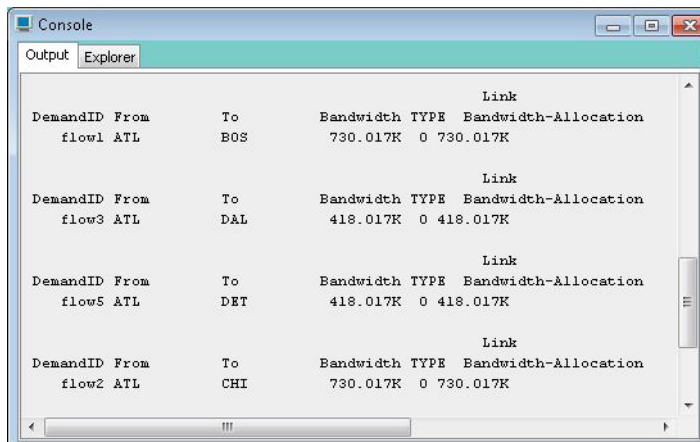


Figure 46: Console

Performing a Path Analysis

1. To perform a path analysis and check if there is enough bandwidth to route a circuit between a given pair of nodes, select **Network > Path & Capacity > Path**. The **Demand Path** window will be displayed.

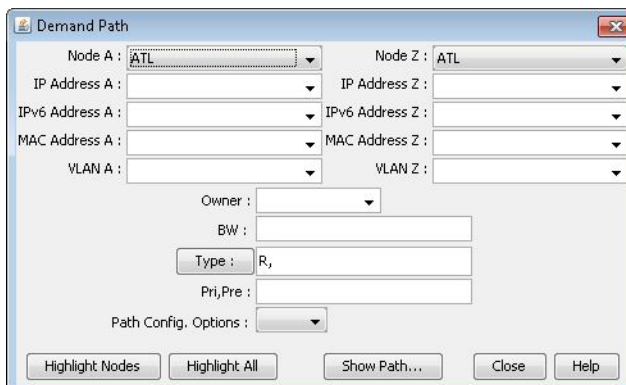


Figure 47: Demand Path Window

2. Select a source and destination node from the drop-down selection boxes. You can also select these nodes from the map by clicking on them. Specify any other parameters you want and then click **Show Path**. If a path is found, it is displayed on the Map and a **Paths** window will appear showing the hop-by-hop path from node A to node B. The console also provides information concerning capacity, route cost and maximum path bandwidth.

Or you can click on the “P” icon on the map toolbar and then click on the source and destination nodes.

3. The Paths window can be customized to show additional columns by right-clicking on the column header and selecting **Table Option**.

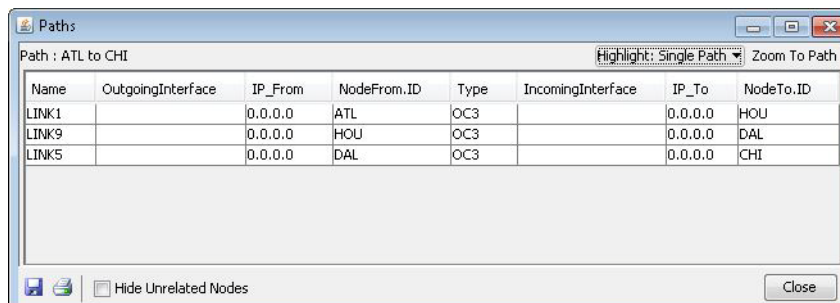


Figure 48: Paths Window

The results will give you information about where that demand would be placed. Additional information displayed in the console includes the **Route Cost** (a sum of the administrative weights* of the path that is chosen) and the **Max Path Bw**, or the maximum demand bandwidth that could be successfully placed on the path. The Max Path Bw is the least amount of available bandwidth along the links in the path.



Informational Note: For IP networks, the routing method (for example, OSPF, ISIS) is set through **Tools > Options > Design > Path Placement**. For ATM networks, the you may specify different routing methods for different QoS through **Tools > Options > Design > Path Placement > Partition**. If the **Routing Method** is set to "Constant Distance", then the **Route Cost** will display the number of hops for the path.

Note that the **Routing Method** is initialized to the default setting used by the hardware. Though you may specify different routing methods for different service classes, this may not be supported in the hardware. However, this flexibility is provided as it allows you to calculate desirable paths to be configured in the network.

4. For an ATM network, you can additionally specify the **Quality of Service** by clicking on the "**Type**" button and then selecting the desired radio button (**CBR**, **RT**, **NRT**, and so on.). For ATM, if you specify a **BW** value (for example "2M") in the demand path options in Figure 47, the Console and Map will indicate whether your demand fails to be placed because of insufficient available bandwidth.

Note that this is a temporary what-if study. No changes will be made to your network model.

5. From the **Network > Path & Capacity** menu, you can also select:
 - **Diverse Path** to find out if there is enough bandwidth to route two circuits in node-disjoint (diverse) paths between a given pair of nodes. If node-disjoint paths are not found, edge-disjoint paths will be searched for. If edge-disjoint paths are not found, the program engine will search for paths with a minimum number of common nodes or edges.
 - **Site Path** to check if there is enough bandwidth to route a path between two selected sites.
 - **Site Diverse Path** to check if two node-disjoint (diverse) paths exist between sites.
 - **Equivalent Path** (routers only) to see an equivalent path list between two nodes.

Viewing Aggregate Demand Information

1. To view the total bandwidth between pairs of nodes A and Z in the network sorted by bandwidth in descending order, select **Network > Elements > Demands**. Right-click a demand and select **Aggreg Demands** to get the following window.

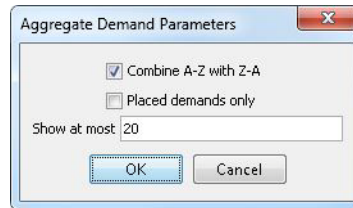


Figure 49: Aggregate Demand Parameters

2. Check the **Combine A-Z with Z-A** checkbox to use the higher of the two directions doubled as an approximation of the total bandwidth of demands in the A to Z and Z to A directions combined. (Alternatively, you could also uncheck it if you want to see the exact summation of the bandwidth of demands in the A to Z direction and the bandwidth of demands in the Z to A direction. In the latter case, two rows are displayed for each node pair rather than one.)
3. Check the **Placed demands only** checkbox to only consider the bandwidth of demands that have been placed in the current model of the network.
4. Change the **Show at most** text box to 10 to view the 10 node pairs with the largest bandwidth. Then click “**OK**” to view the following window.

This table has a row for each node pair, showing A, Z, and the total bandwidth. It is sorted by bandwidth, with the largest at the top. To sort by a different field or to change between ascending and descending order, click on the corresponding column header.

5. Click **Highlight All** to highlight all the node pairs shown in the table.

Adding Nodes, Links, and Demands

This chapter describes how to add nodes and links to a network model through the IP/MPLSView client.

When to use

Use these procedures when you want to create your own network model using the IP/MPLSView client. Alternatively, you can create your own network model by creating a nodes text file (*muxloc*) and links text file (*bblink*). Or, for a router module, you can import configuration files into your network as discussed in the Router Feature Guide for IP/MPLS, and the corresponding network files will automatically be generated.

Prerequisites

Before beginning this task, you must have created a specification file as described in Chapter 5, Creating and Editing a Network Model or opened a specification file as described in Chapter 4, Opening and Closing a Network Project. You can then continue with these procedures to modify a specification file.

Related Documentation

For an overview or detailed description of each window, refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Recommended Instructions

Following is a high-level sequential outline of the specification file creation process and recommended procedures.

1. Create the specification file to be updated as described in step 1 on page 67.
2. Add three nodes as described in step 1 on page 68.
3. Add three links to form a triangle-shaped network as described in step 1 on page 69.
4. Add a single demand and then update the network state as described in step 1 on page 71.
5. Add multiple demands by specifying source and destination nodes and the bandwidth of the demand and then update the network state as described in step 4 on page 72.
6. Switch back to view mode and save your changes as described in step 1 on page 73.
7. View the resulting text files as described in step 3 on page 73.

Detailed Procedures

Adding a Node

1. In the **Network Info** window **Nodes** tab, click the **Add** button to add a new node to the network. The **Add Node** window will appear.

The screenshot shows the 'Add Node' dialog box with the 'Properties' tab selected. The 'Apply Template' dropdown is set to 'Default'. The 'Properties' section includes fields for ID, Name, IP Address, IPv6, Hardware (set to 'ALL'), OS, L2SW (set to 'false'), and a Comment field. The 'Location' section has a 'Choose Location Using' section with radio buttons for 'Mouse' (selected), 'City/Country', 'NPANXX', and 'Lat/Lon'. Below this, there are dropdowns for Country (set to 'US UNITED_STATES') and City, a 'Search' button, and input fields for NPANXX / Lata (set to '999999'), Lat / Lon (set to '0.0'), and Site ID (set to 'NONE'). At the bottom are 'Add...', 'Reset', 'Close', and 'Help' buttons.

Figure 50: Add Node Window

2. The only required field is the **"ID"** field. Type in the ID **"N1"**, name **"node1"**, and an appropriate hardware type for your network.
3. By default, **"Choose Location Using: Mouse"** will be selected." Click over to the topology window or select **Windows>Map (Standard)** and you will see that the cursor will become a cross-hair on the map. Click on the desired location for the node. Notice that the coordinates are automatically inserted in the **Add Node** window. In the **Add Node** window, click on **"Add."** The node gets added to the location you specified using your mouse.
4. The **Add Node** window will remain open. This time, enter the ID **"N2"** and name **"node2"**. In the **Location** section, click on the **NPANXX** checkbox and specify the NPANXX data in the NPANXX textbox. Click on **"Add"**. The node gets added to the location with the NPANXX you specified.

5. Add a third node with ID as "N3" and name "node3". In the **Location** section, check the **City,Country** checkbox to specify location by city and country. Select **US UNITED_STATES** from the **Country** drop-down menu. Then type in "Chicago" in the **City** textbox and click "**Search**." It will come up with a selection of cities that include the name "Chicago". Select the one that says "CHICAGO_IL." Click on "**Add**." The node gets added to the city you specified.



Informational Note: If the city you want to enter in does not exist, refer to "Customizing the Location Database" of this chapter for details on how to add locations to the database. If you have the CLLI codes and want to convert them to npa nxx, refer to "Deriving npa nxx from CLLI codes using wanpricer*".

The node ID for an added node must be unique, while the node name does need not be unique. You can modify nodes by selecting its entry in the **Nodes** tab and click the **Modify** button.

6. Click on "**Close**" to close the **Add Node** window.
7. You should have 3 nodes on the Map.

Adding a Link

1. Now that you have added some nodes, the next step is to add links between them. In the **Network Info** window **Links** tab, click on the "**Add**" button to add a new link to the network. The **Add Link** window will appear.

Figure 51: Add Link Window

2. Enter in the link name "Link_N1_to_N2" into the **Name** field and select "T1" for **Trunk**. Leave the other properties as they are.
3. Check the checkbox labeled "**Auto Add on Mouse Clicks**" in the top right corner of the **Add Link** window.
4. Move the **Add Link** window and/or Map window around so that the nodes N1 and N2 are visible.

5. Put your mouse over the Map and you will see that the cursor will become a cross-hair on the map. Click on node N1 for your first endpoint. Then click on node N2 as your second endpoint. The map's bottom bar will display a brief message that a link was added. A blue link will be drawn between the two endpoints.
6. Move the **Add Link** window back into view. Now try to add a second link named "Link_N2_to_N3" between nodes N2 and N3. This time deselect the "**Auto Add on Mouse Clicks**" checkbox. To specify the location, click on the **Location** tab.

The screenshot shows the 'Add Link' dialog box with the 'Location' tab selected. The 'Apply Template' dropdown is set to 'Default'. The 'Auto Add on Mouse Clicks' checkbox is unchecked. The 'Name' field is empty. The 'Trunk' dropdown is set to 'OC12', 'Vendor' to 'DEF', and 'Cost' is empty. The 'Fixed' checkbox is unchecked, and 'CanFail' is checked. The 'Avail Facility', 'Misc', and 'Comment' fields are empty. The 'BW', 'Ovhd', 'Delay', 'Metric', 'Tunnel Metric', and 'Layer' fields are empty. The 'Oper Status' dropdown is set to 'Active', 'Admin Status' to 'Unknown', 'Geo Dist' is empty, and 'Routing Instance' is set to 'NONE'. The 'Add...' button is highlighted.

Figure 52: Add Link Window Location Tab

7. Click on the "**NodeID**" radio button to choose the location using node IDs. Then select N2 for **Node A** and N3 for **Node Z**. Click "**Add**."
8. Click back to the **Properties** tab. Add a third link named "Link_N3_to_N1" using the same method used for Link_N2_to_N3. After completing this addition, click "**Close**" to close the **Add Link** window.
9. You should have the 3 nodes connected on the Map forming triangle.

Adding Demands

1. Finally, you can add demands to the network. In the **Network Info** window **Demands** tab, click the **Add > One Demand** button to open the **Add Demand** window. The **Add Demand** window will appear.

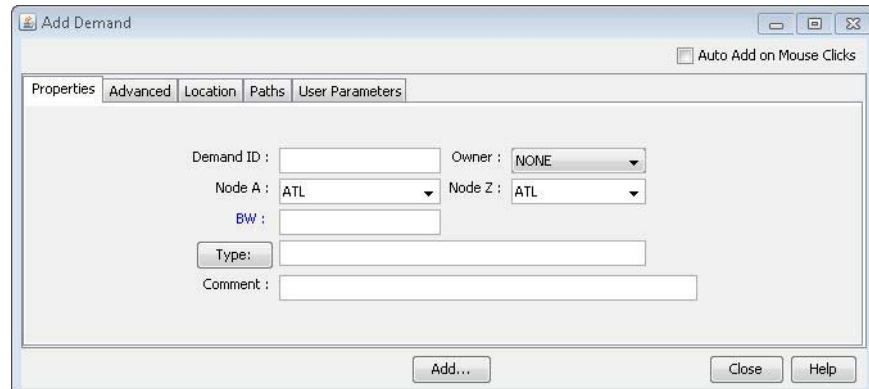


Figure 53: Add Demand Window

2. For **Demand ID**, the identifier for the flow or circuit, type in "Dmd_N1_to_N2." Then select N1 from the **Node A** (Origin) drop-down menu and N2 from the **Node Z** (Destination) drop-down menu.

For **BW** (bandwidth), put in 1M (for 1Mbps) for the A to Z direction only, and for **pri,pre**, put in 2,2. Leave default settings for the rest of the items. Click "**Add**" to add the demand. A pop-up note will notify you that the demand has been added. Close the **Add Demand** window.
3. Your Map will now have a link with utilization in one direction and colored according to your utilization legend. The color originating from N1 indicates that the demand is placed in the direction from N1 to N2 rather than the reverse direction.

- Next, try adding multiple demands at one time. Click the **Add > Multiple Demands** to bring up the **Add Multiple Demands** window.

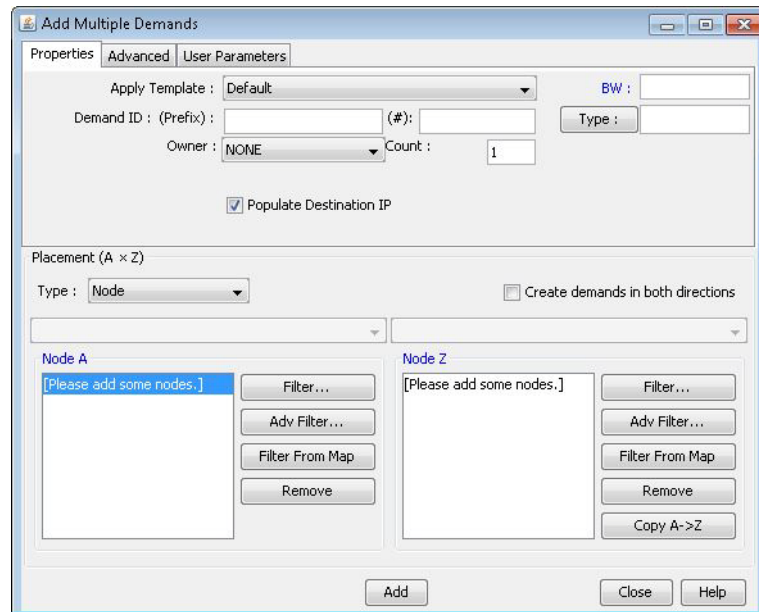


Figure 54: Add Multiple Demands Window

- Now this tutorial will guide you in adding a demand for each source-destination node pair. In this window, specify “MyDmd” in the **Demand ID : (Prefix)** text box, and “1” after **(#)**. This will cause your demand IDs to be automatically generated, starting from MyDmd_1 and counting up.
- For the bandwidth, use 750K.
- Under the **Placement (A x Z)** section, press the “**Filter**” button in the **Node A** section to filter all the source nodes for the multiple demands. The **Find Nodes** window will appear.

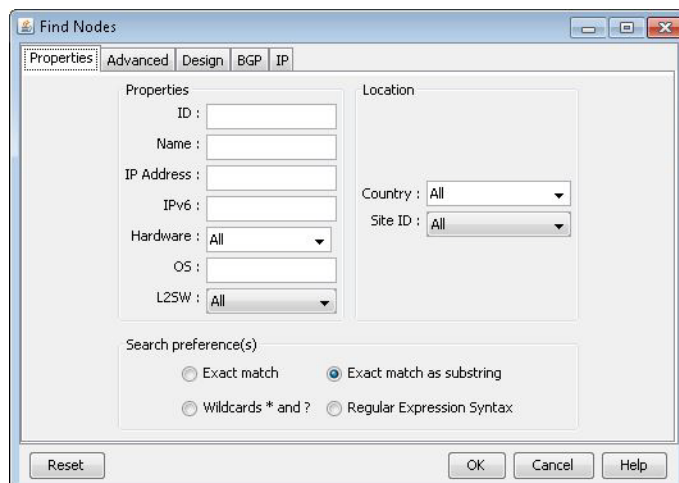


Figure 55: Find Nodes Window

8. Leave the fields empty and click **“OK”** to fetch all nodes. Nodes N1, N2, and N3 will be listed under Node A as the source nodes.
9. For Node Z, click on the **“Copy A->Z”** to use the same nodes as the destination nodes.
10. Click **“Add”** to get a confirmation box notifying you that the demands will be added. Click **“Yes”** and when the **Info** dialog box is displayed, then click **“OK”**.
11. Your Map should have utilization on all links and colored according to your utilization legend.
12. In the **Utilization Legends** pane, try moving the color dividers around to distinguish different link utilizations.
13. To see how each demand was placed, go to the **Network Info** window **Demands** tab.

Saving Network

1. Finally, to save your changes select **File > Save**. A default directory will be provided which can be changed.
2. Choose a runcode. If the runcode already exists in the directory of choice, this will overwrite the existing files.
3. After you have saved the network files, you can view them in **File Manager** by navigating to the saved directory. To view the spec as a text file, right-click on the file name and select **Edit**. To view all other files, double-click the file.

Sample Files

Sample files created from the exercise.

spec.triangle3

```
datadir = .
runcode = triangle3
# ratedir = /u/wandl/db/rates/default
muxloc = muxloc.triangle3 # mux location file
nodeparam = nodeparam.triangle3 # mux type and constraints file
bbblink = bblink.triangle3 # backbone link file
graphcoord = graphcoord.triangle3 # defines graphics coordinates of nodes
graphcooraux = graphcooraux.triangle3 # aux graphics coordinates of nodes
demand = demand.triangle3 # circuit definition file
dparam = dparam.triangle3 # parameters
usercost = none # user defined link cost file
linkdist = none # user defined link distance file
nodeweight = nodeweight.triangle3 # user defined node weight & bandwidth limit
```

dparam.triangle3

```
hwvendor=GENATM
```

muxloc.triangle3

```
N1          NODE1 732 271  US 482459N  0775825W
N2          NODE2 732 868
N3          NODE3 773 223  US 415042N  0874149W
```

bblink.triangle3

#LinkName	Node	Node	Vendor	#	BwType	Misc
LINK_N1_TO_N2	N1	N2	DEF	1	T1	
LINK_N3_TO_N1	N1	N3	DEF	1	T1	
LINK_N2_TO_N3	N2	N3	DEF	1	T1	

demand.triangle3

Dmd_N1_to_N2	N1	N2	1000000	R,A2Z,NRT	02,02	LINK_N1_TO_N2
MyDmd_1	N2	N3	750000	R,A2Z,NRT	07,07	LINK_N2_TO_N3
MyDmd_2	N2	N1	750000	R,A2Z,NRT	07,07	LINK_N1_TO_N2
MyDmd_3	N3	N2	750000	R,A2Z,NRT	07,07	LINK_N2_TO_N3
MyDmd_4	N3	N1	750000	R,A2Z,NRT	07,07	LINK_N3_TO_N1
MyDmd_5	N1	N2	750000	R,A2Z,NRT	07,07	
MyDmd_6	N1	N3	750000	R,A2Z,NRT	07,07	LINK_N3_TO_N1

graphcoord.triangle3

```
worldcoord
window -10211 -15892 17271 17271
mktp 3 # marker type
#node npa nxx latitude longitude or
#node npa nxx graph_v graph_h
N1 999 999 -67.0119 -95.1504
N2 732 868 32.0560 -32.2654
N3 219 545 33.1463 -166.3667
```

nodeparam.triangle3

```
# Format Example:
# N001 MUX
N1 MUX
N2 MUX
N3 MUX
```

Deriving npa nxx from CLLI codes using wanpricer*

Wanpricer is a special feature requiring a separate license. Please contact your Juniper representative for more information.

The following are the instructions to convert CLLI codes to npa nxx:

1. Create a file with the list of CLLI codes, one entry per line.
2. Next, run the program "/u/wandl/bin/wanpricer". Answer "n" when asked to read/create a specification file. Then select the following menu options:
3. From the WANPricer Main Menu, select **D. Domestic U.S. Tariff & POP Homing**.
4. From the Domestic U.S. Tariff & POP Homing Menu menu, select **1. Database Information**.
5. From the Database Information menu, select **6. VH/LL/CLLI to NPA Conversion**.
6. From the VH/LL/CLLI to NPA Conversion Menu, select **5. CLLI to NPANXX File Conversion**.
7. Next, enter the file name in which to save the results. Press enter or quit until you get out of the program.
8. The resulting file has a npa nxx listed for each CLLI code, which can be specified in the muxloc file with the following format: NodeID Name npa nxx

9. Note that the NodeID should match with the NodeID used in the rest of the network project. When loading up the network, the npa nxx will be used to position the nodes in the right location.
10. Upon loading the network with this updated npa nxx information, verify that the nodes are in the correct locations.

Customizing the Location Database

If a country or city is missing from the database, you can add it using the procedures in the following sections.

Adding a Country

To add a country, edit the `/u/wandl/db/country/countryloc.txt`

```
#code av_lat av_lon max_lat max_lon min_lat min_lon exchange name continent
```

Column	Description
code	2 letter country code
av_lat and av_lon	Average latitude and longitude
max_lat and max_lon	Maximum latitude and longitude
min_lat and min_lon	Minimum latitude and longitude
exchange	Long version of name
continent	A: Asia N: North America E: Europe S: South America F: Africa O: Ocean (Use dash '-' if uncertain)

Creating a City Database for a Country

1. Create a file in `/u/wandl/bin` (or `$WANDL_HOME/bin` if `/u/wandl` is not found) with the name `[CountryCode].lat`, substituting `[CountryCode]` with the two-letter countrycode. Enter into the file one city per line using the following fields:

```
2letterCountryCode Latitude Longitude Gateway City
```

(For the gateway field, enter "G" for gateway or "-" if not a gateway.)

Example CN.lat:

```
CN 51.1 -114.017 G Calgary
CN 53.5667 -113.517 G Edmonton
CN 55.1833 -118.883 G GrandePrairie
CN 52.8833 -118.067 G Jasper
```

2. Run `/u/wandl/bin/country2obj` (or `$WANDL_HOME/bin/country2obj` if `/u/wandl` is not found) in the `/u/wandl/bin` directory.

```
country2obj [2letterCountryCode]
```

Example:

```
$ cd /u/wandl/bin
$ country2obj CN
```

-
3. Make a backup of the current obj file in /u/wandl/db/country that you want to replace. Then move the new .obj file to /u/wandl/db/country (or \$WANDL_HOME/db/country if /u/wandl is not found).



Informational Note: By moving an obj file, you will be replacing the previous country obj file, not adding onto it.

4. You may need to close your currently opened networks and reopen them for the changes to take effect.

Modifying Nodes, Links and Demands

This chapter describes how to modify nodes and links to a network model through the IP/MPLSView client.

When to use

Use these procedures when you want to modify your own network model using the IP/MPLSView client. Alternatively, you can modify your own network model by editing network files like the node file (muxloc) and link file (bblink) according to IP/MPLSView file format.

Prerequisites

Before beginning this task, you must have created a specification file as described in Chapter 5, Creating and Editing a Network Model or opened a specification file as described in Chapter 4, Opening and Closing a Network Project. You then should continue with these procedures to modify a specification file.

To follow along with the first part of the current tutorial, you should have created “spec.triangle3” as explained in Chapter 9, Adding Nodes, Links, and Demands.

Related Documentation

For an overview of IP/MPLSView or for a detailed description of each window, refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Recommended Instructions

Following is a tutorial to help the user learn different methods to modifying nodes, links, and demands.

- Modify nodes, links, and demands.
- Deleting nodes, links, and demands.

Detailed Procedures

Modifying Nodes

1. Open the “spec.triangle3” network that was created in Chapter 9, Adding Nodes, Links, and Demands, or open another network with existing network elements.

2. In the **Network Info** window **Nodes** tab, select an entry from the table, and then click the **Modify** button to open the **Modify Nodes** window.

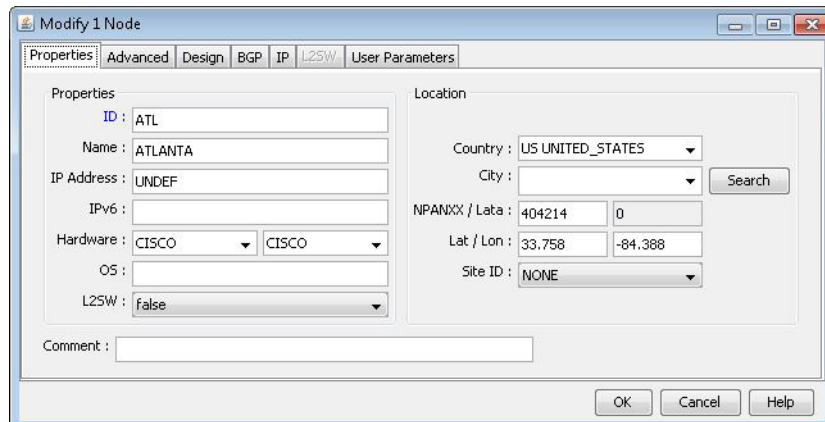


Figure 56: Modify Single Node Window

3. If you want to modify multiple nodes simultaneously, select multiple entries with **<Ctrl>**-click or **<Shift>**-click, and then click on the **Modify** button. Any field changed will be applied to all the selected nodes. Fields left blank will keep their existing values.

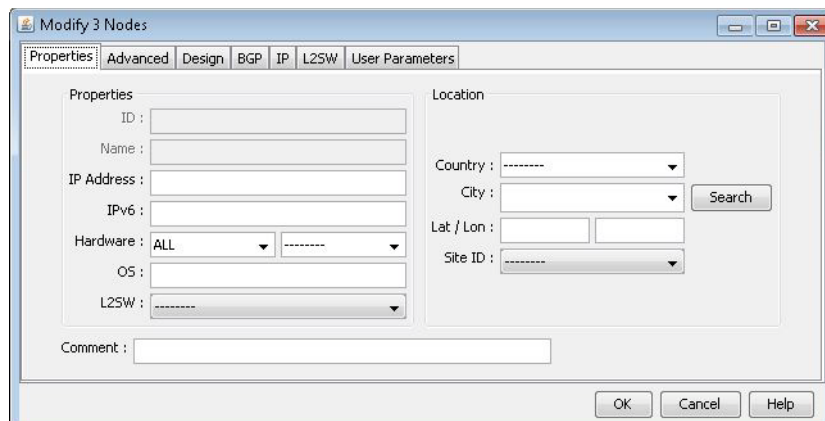


Figure 57: Modify Multiple Nodes Window

4. Type in a comment in the **Comment** text box. Then, click the **OK** button to submit the changes. Click **Yes** to confirm that you want to make the modification. Or, for a multiple node modification, you may wish to click **Yes to All**.
5. Verify the changes you just made in the **Nodes** tab. You can check the Comment field in the detailed information tabs. Or you can add the Comment field to the column headers.

Modifying Links

1. In the **Network Info** window **Links** tab, select an entry from the table, and then click the **Modify** button to open the **Modify Links** window

Figure 58: Modify Single Link Window

2. If you want to modify multiple links simultaneously, select multiple entries with **<Ctrl>-click** or **<Shift>-click**, and then click on the **Modify** button. Any field changed will be applied to all the selected links. Fields left blank will keep their existing values.

Figure 59: Modify Multiple Links Window

3. Change the Trunk type. Then, click the **“OK”** button to submit the changes. Click **“Yes”** to confirm that you want to make the modification. Or, for a multiple node modification, you may wish to click **“Yes to All”**.
4. Verify the changes in the **Links** tab in the Type column.

The link colors may turn different colors to indicate the change in utilization to the new trunk type.

Modifying Demands

1. In the **Network Info** window **Demands** tab, select an entry from the table, and then click the **Modify** button to open the **Modify Demands** window.

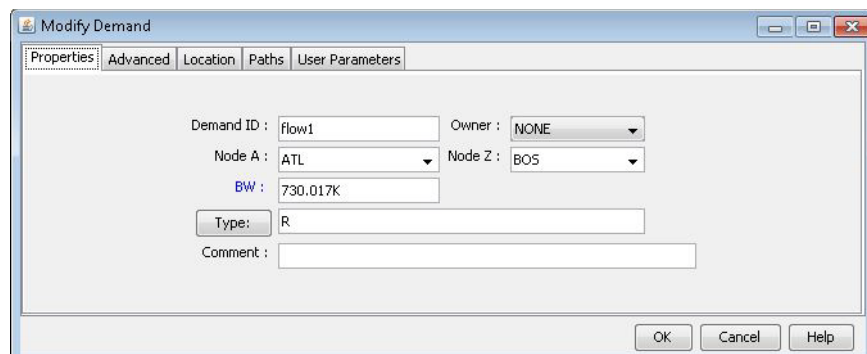


Figure 60: Modify Single Demand Window

2. If you want to modify multiple demands simultaneously, select multiple entries with <Ctrl>-click or <Shift>-click, and then click on the **Modify** button. Any field changed will be applied to all the selected demands. Fields left blank will keep their existing values.

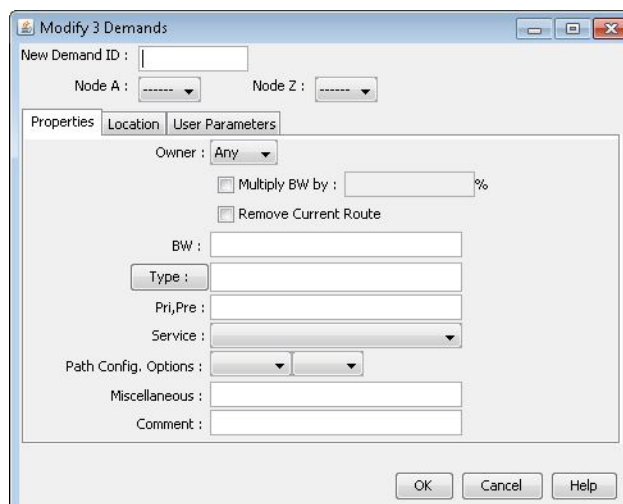


Figure 61: Modify Multiple Demands Window

3. Click the **Search by Property** button to open the **Find Demands** window. Filter the list for only placed demands from a single node, and then click **OK** to view the matched demands.

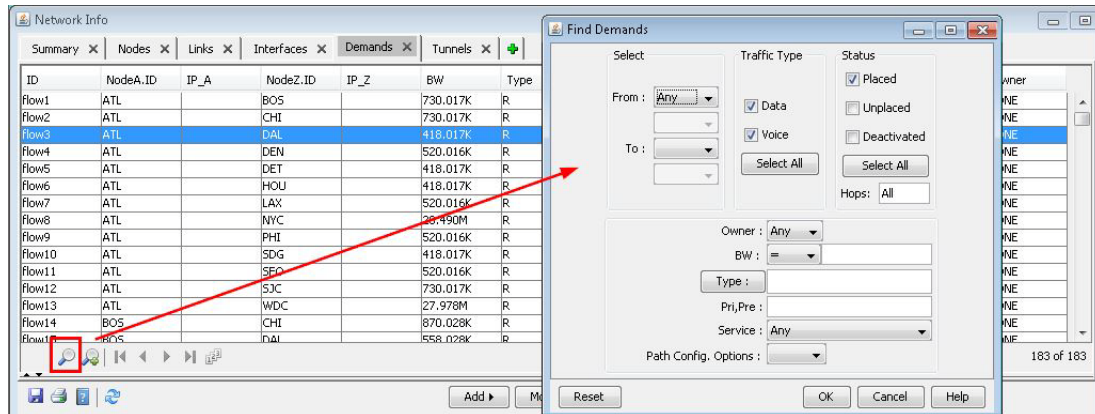


Figure 62: Find Demands Window

4. Only the matching search entries are displayed in the **Demands** tab.
5. Click the **Modify > All Entries** button to modify all demands that meet the search criteria.
6. In the **BW** field, enter in a new value and click **OK.**
7. Verify the changes in the **Demands** tab in the BW column.

Deleting Nodes, Links, and Demands

1. Nodes, Links, and Demands can be deleted in the **Network Info** window in their respective tabs.
2. Select an entry or multiple entries and click the **Delete** button.
3. When an element is deleted, it may delete additional elements associated to it. For example, if a node is deleted, its connecting links will be deleted as well.
4. Elements can also be deleted from the Map by right-clicking on the element and choosing **Delete**.

Modifying Sites and Owners

This chapter is a continuation of the modification features. It describes how to add sites and owners and specify which nodes belong to them. It also describes how to set the admin weight for links and create and specify different service types for demands.

When to use

Use these procedures after creating the muxloc file to perform further modifications on your network model.

Prerequisites

Prior to beginning this task, you must have created a specification file as described in Chapter 5, Creating and Editing a Network Model or opened a specification file as described in Chapter 4, Opening and Closing a Network Project. You then should continue with these procedures to modify a specification file.

Related Documentation

For an overview of IP/MPLSView or for a detailed description of each window, refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Recommended Instructions

Following is a high-level sequential outline of the network modification process and recommended procedures.

1. Create a site and assign nodes to the site: Creating a Site on page 84.
2. Create an owner and specify that owner for a subset of the demands: Creating Owners on page 86.
3. Add, modify, or delete a new Admin Weight/Metric rule for links: Admin Weight/Metric Rules on page 88.
4. Add, modify or delete a Service Type: Service Type on page 90.

Detailed Procedures

Creating a Site

Sites are user-defined groups of nodes. In IP/MPLSView, you should define sites if:

- You wish to perform Site Failure resiliency tests, in which the network is tested for survivability in the event that an entire site (all the nodes in the site) go down. For more information, see the section on failure simulation scripts in Chapter 15, Simulation as well as the simulation chapter in the IP/MPLSView Java-Based Graphical User Interface Reference.
- You are performing a design and want to ensure that links that are purchased between certain groups of nodes will be considered free. In IP/MPLSView, links between nodes in the same site are considered “in-house” and incur no monthly charge. Creating sites will encourage IP/MPLSView to purchase intra-site links, because there is little impact on the overall cost of the network design. Note that there are other ways to achieve the same effect. For more on design, see Chapter 14, Design.



Informational Note: In this section, you can use the specification file located in `/u/wandl/sample/IP/fish/`.

1. In the **Network Info** window, click the plus icon to access **Sites** and additional network element options.

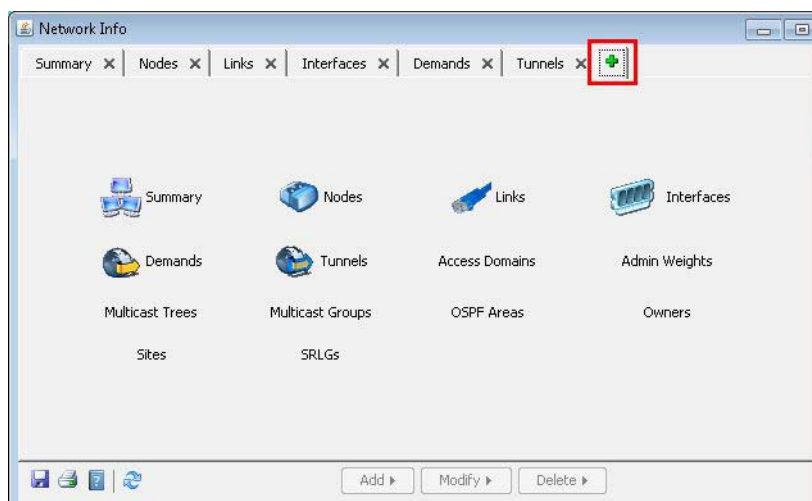


Figure 63: Network Info Additional Elements

- Click on **Sites** and then click **Add > One Site** to create a new site.

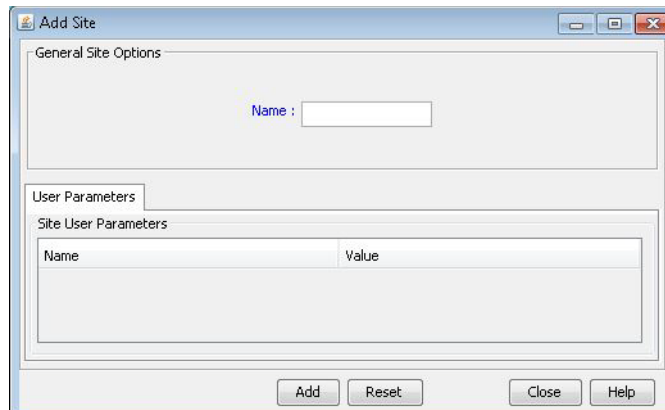


Figure 64: Add Site Window

- In the **Add Site** window, enter a site name. In this example, we will use the site name "WEST". Click **Add** to add the new site.
- Use the **Select Tool** to select four nodes on the west side of the Map.
- Right-click on one of the nodes and select **Modify > Selected Nodes**. This opens the Modify Multiple Nodes window where you can change the Site ID to "WEST".

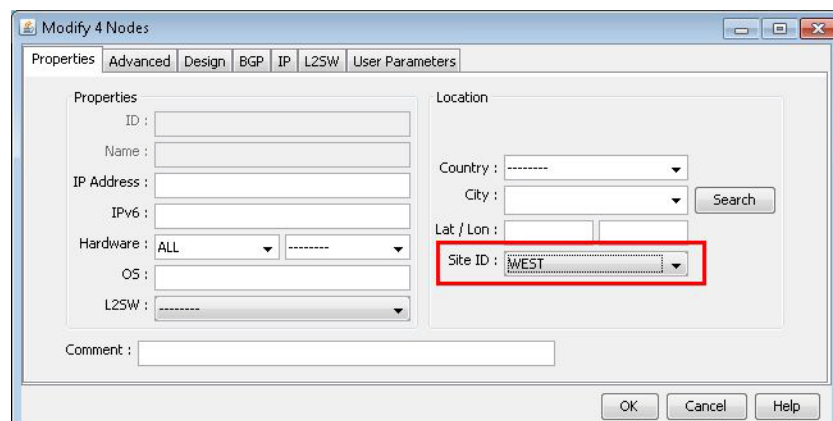


Figure 65: Modify Site for Selected Nodes

- Click the **OK** button to submit the changes. Click **Yes to All** in the confirmation dialog box to apply the change all the selected nodes.
- Use the same procedures to create another site named "EAST" and add three eastern nodes into this site.

The **Sites** tab should now display the “WEST” and “EAST” sites.

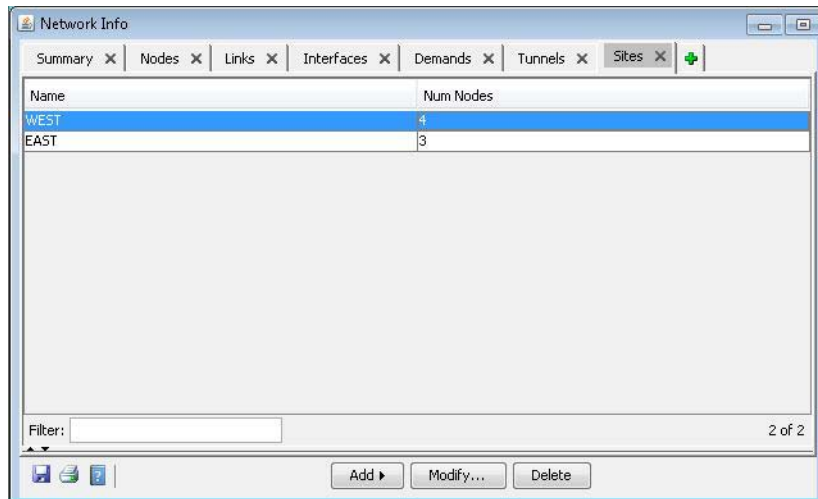


Figure 66: Sites Tab

Creating Owners

Owners offer a flexible way of organizing your demands. You can associate owners with the demands belonging to your different network customers. Or you can associate owners with demands belonging to different network applications, different internal departments, and so on. Using owners can help with reporting because many of the demand-related reports can be sorted by owner.



Informational Note: In this section, you can follow along using the specification file in /u/wandl/sample/IP/fish.

1. In the **Network Info** window, click the plus icon to access **Owners** and additional network element options.
2. Click on **Owners** and then click **Add** to open a text box to enter the Owner name.

3. Specify the owner name in the text field and click the plus icon to add the entry.

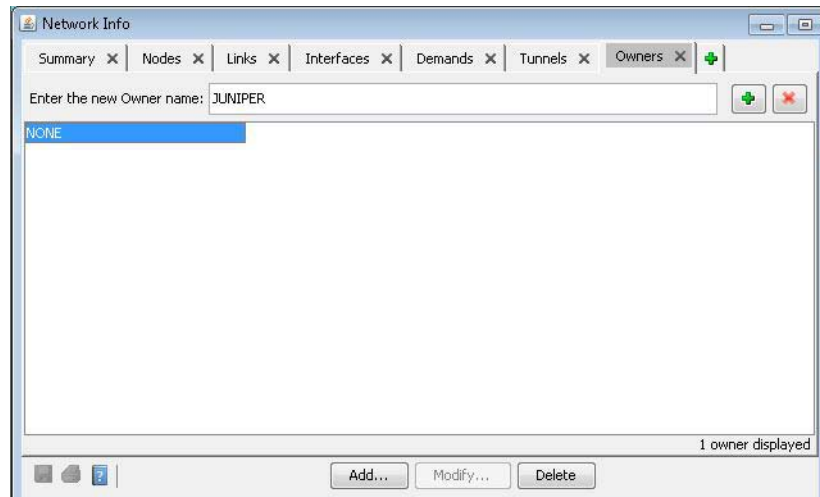


Figure 67: Owners Tab

4. The next step is to select a number of demands and associate them to an owner. In the **Demands** tab find all the demands that originate from one node, such as ATL. When you have selected those demands, click **Modify > Selected**. Set the Owner and click **OK**.

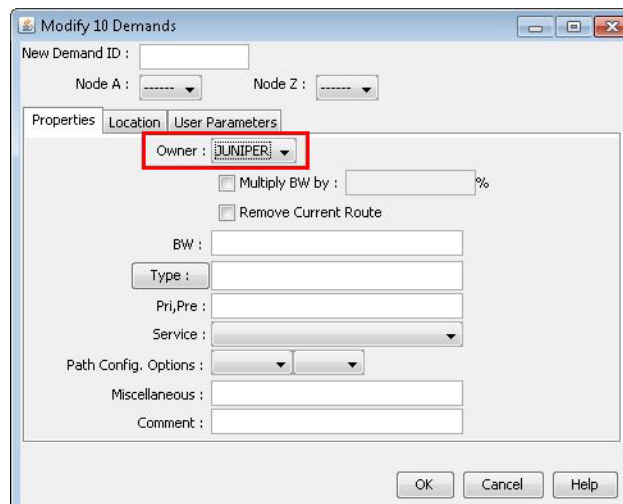


Figure 68: Modify Demands to Set Owner

5. In the **Owners** tab, right-click an owner and select **View Demands** to view the demands that are owned by the selected owner.

ID	NodeA.ID	IP_A	NodeZ.ID	IP_Z	BW	Type	Pri
flow1	ATL		BOS		730.017K	R,PROV_ST...	02
flow2	ATL		CHI		730.017K	R,PROV_ST...	02
flow3	ATL		DAL		418.017K	R,PROV_ST...	02
flow4	ATL		DEN		520.016K	R,PROV_ST...	02
flow5	ATL		DET		418.017K	R,PROV_ST...	02
flow6	ATL		HOU		418.017K	R,PROV_ST...	02
flow7	ATL		LAX		520.016K	R,PROV_ST...	02
flow8	ATL		NYC		20.490M	R,PROV_ST...	02
flow9	ATL		PHI		520.016K	R,PROV_ST...	02
flow10	ATL		SDG		418.017K	R,PROV_ST...	02

Figure 69: Demands Belonging to the Selected Owner

6. Right-click on a demand entry and select **Highlight All** to view the end points of those demands.

Admin Weight/Metric Rules

The admin weight tool allows the user to specify the rules that should automatically be used to calculate the admin weights/metrics for links which do not already have a specific admin weight/metric specified in the *bblink* or the *linkdist* file. This weight is then taken into consideration for routing, when the protocol is looking for the shortest path to take.

Specifying Link Characteristics

The user can specify which links to apply an admin weight formula to by matching those links with certain link characteristics that must be met. Links can be matched against characteristics such as protocol type, hardware type of the link end nodes, and trunk type. The user can also restrict an admin weight rule to apply only to intra-site or intra-peer group links. For these options, “**Don’t Care**” indicates that a rule should be applied regardless of whether the link is intra-site or intra-peer.

Specifying a Formula Using Fixed and Variable Weight

Given a formula for links in a particular category, the default weight for such a link is calculated to be a fixed weight plus the link’s airline distance times a variable weight.

$$\text{default weight} = \text{fixed weight} + (\text{link's airline mileage}) * \text{variable weight}$$

Setting Multiple Formulas

Note that the user can specify several different formulas, each to be applied to links with different characteristics. In the case that a link fits two sets of characteristics for which two different formulas are defined, the formula associated with the more specific criterion will be used.

For example, suppose you specify a fixed weight of 100 and variable weight of 0 for links of any protocol (protocol set to “ALL”) and fixed weight of 200 and variable weight of 0 for links of protocol PNNI. In this case, for PNNI links, you would apply the latter, more specific formula. For non-PNNI links, you would apply the former formula since that is the only one applicable of the two.

1. In the **Network Info** window, click the plus icon to access **Admin Weights** and additional options.
1. Click the **Add** button.
2. Set the protocol for the type of link, the type of trunk, the hardware type of the node A and node Z, the hierarchy level of node A and node Z, and whether the weight is for links connecting nodes in the same site. Specify a fixed weight and a variable weight.

Figure 70: Add Admin Weight Window

3. When all the values have been set, click **OK**.
4. These settings will be displayed in the **Admin Weights** tab.
5. To modify any record(s), select and highlight the record(s), then click on the **Modify** button. A Modify Admin Weight window will appear allowing you to change any values in the window. When you are finished with the modification, click **OK**.
6. To delete any record(s), select the record(s) in the Admin Weights table and click on the **Delete** button.



Informational Note: When you save the network environment, a file called “admincost.runcode” will be created. This file can also be created manually and specified in the specification file. Refer to the chapter on Control Files in the “File Format Reference for IP/MPLSView” for more details.

7. To view link metrics on the map, right-click on the Map and select **Labels > Link Labels > Show Link Metrics**.



Informational Note: Keep in mind that the weights defined in **Admin Weight** are only used when a more specific distance metric is not specified in the bblink or linkdist files.

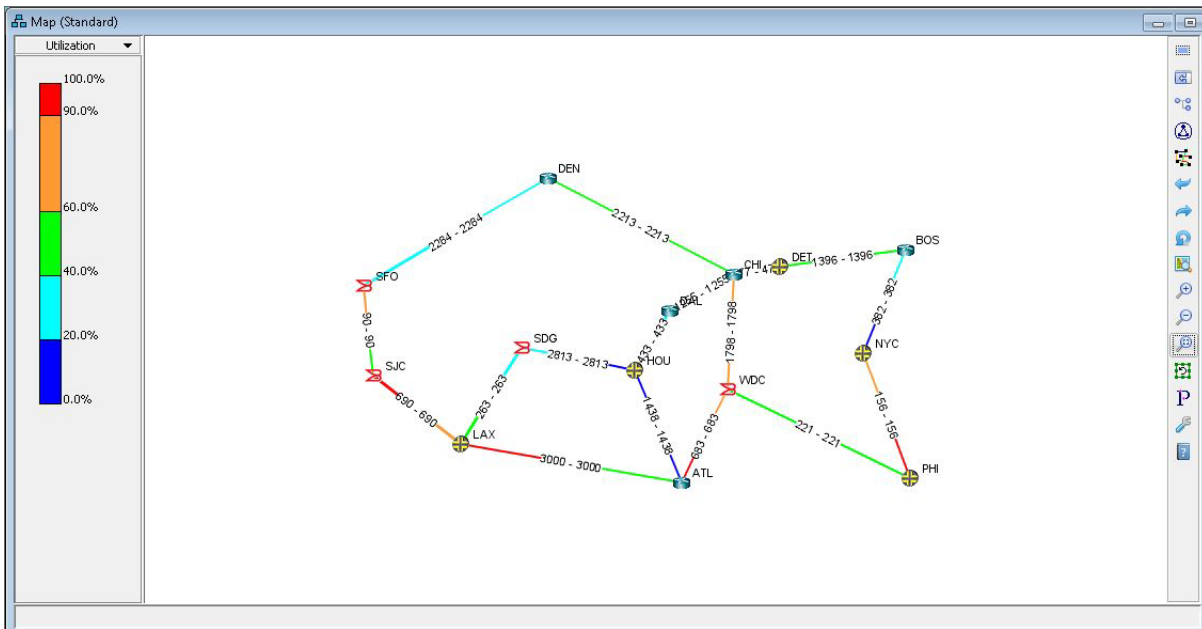


Figure 71: Link Distances Displayed on Topology Map

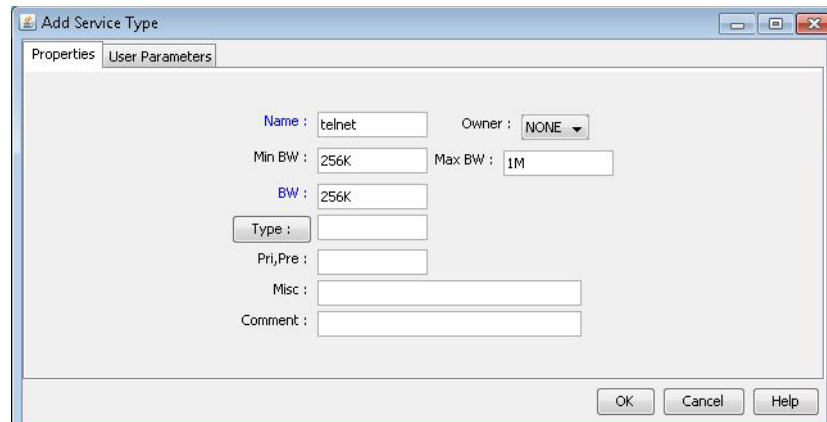
Service Type

Service type is a way of describing traffic types for demands. When creating or modifying demands, the service type can automatically fill in the fields for the specified service type of the new demand.

It also allows for capabilities that were not allowed in the demand template since it is not only a template but also a demand field. For example, you can search for demands by the service type field and you can add single demands with a service type.

1. Select from the main menu **Network > More > Service Type**.
2. In the **Service Type** tab click **Add**.
3. Add a new service type named "telnet", with 256K as the minimum bandwidth, 1M as the maximum bandwidth, and the bandwidth from A-Z as 256K.

- Click on the **Type** button to add further specifications such as the Class of Service.



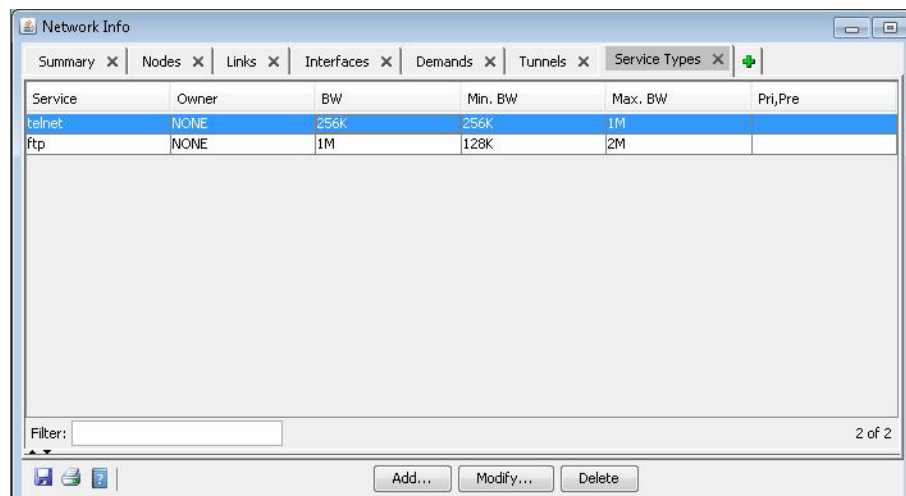
The 'Add Service Type' dialog box has two tabs: 'Properties' and 'User Parameters'. The 'Properties' tab is active. It contains the following fields:

- Name :** telnet
- Owner :** NONE (dropdown menu)
- Min BW :** 256K
- Max BW :** 1M
- BW :** 256K
- Type :** (button)
- Pri,Pre :** (text field)
- Misc :** (text field)
- Comment :** (text field)

At the bottom right are buttons for **OK**, **Cancel**, and **Help**.

Figure 72: Add Service Type

- Click the **OK** button to add the new entry. The new record "telnet" will then appear in the list of service types on the right side of the **Service Types** window.
- Add another service type, "ftp", and enter the fields: Min BW=128K, Max BW=2M, BW=1M, and then click on the **OK** button to close the **Add Service Type** window.
- To modify any record, select and highlight the record and click on the **Modify** button.
- To delete any record, select and highlight the record and click on the **Delete** button.
- To apply the service types to demands, open the **Demands** tab, select the desired demands, click **Add** or **Modify** and set the **Service** in the **Advanced** tab. You will be asked if you wish to populate the fields from the service type demand template.
- If you wish to use the service type only as a template and not as a demand field, you can change the service type back to "NONE" after populating the fields.



The 'Network Info' window shows the 'Service Types' tab. It contains a table with the following data:

Service	Owner	BW	Min. BW	Max. BW	Pri,Pre
telnet	NONE	256K	256K	1M	
ftp	NONE	1M	128K	2M	

Below the table is a 'Filter:' text field. At the bottom right, it says '2 of 2'. At the bottom center are buttons for **Add...**, **Modify...**, and **Delete**.

Figure 73: Service Types Tab

User Parameters

This chapter describes how to use the User Parameters feature in order to provide greater flexibility in characterizing and working with nodes, links, sites, demands, and tunnels in your network.

When to use

Use this feature if you wish to specify custom parameters for your various network elements or wish to perform the validation of links against defined templates.



Informational Note: The tunnel user parameters require a license for MPLS tunnels. Please contact your Juniper representative for more information.

Prerequisites

Prior to beginning this task, you should have started up IP/MPLSView and opened a specification file as described in Chapter 4, Opening and Closing a Network Project.

You should know how to add and modify your network, as explained in Chapter 9, Adding Nodes, Links, and Demands and Chapter 10, Modifying Nodes, Links and Demands.

Related Documentation

For an overview of IP/MPLSView or for a detailed description of each window, refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Outline

- User Parameters Window on page 94
- Modifying User Parameter Values on page 95
- Performing Text Filter on User Parameters on page 95
- Map Labels with User Parameters on page 96
- The Usrparam File Format on page 96

User Parameters

The User Parameters feature allows you to define up to five custom parameters for each type of network element (nodes, links, sites, demands, tunnels). Values can be assigned to these parameters for each individual network element. These defined parameters and values can then be used for several different purposes which will be discussed below.

User Parameters Window

1. From the main menu, select **Network > Elements > User Parameters**.
2. Click on the different tabs to see the defined User Parameters for each network element. User Parameters can be created for **Nodes**, **Links**, **Sites**, **Demands**, and **Tunnels**.
3. Click **Add**.
4. Enter a name of the User Parameter and click **OK**.
5. To rename a User Parameter, select it from the table and click **Modify**.
6. The order of the User Parameters can be rearranged with the up and down arrows.

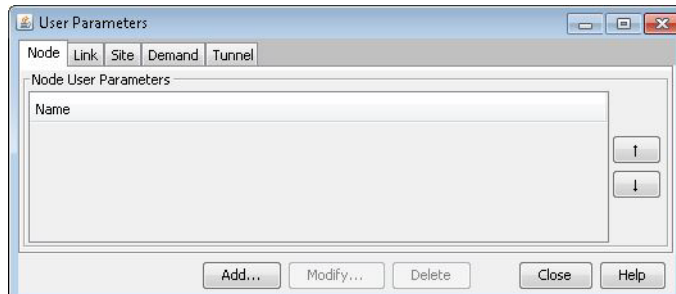


Figure 74: User Parameters Window

Modifying User Parameter Values

You can view the defined User Parameters for each network element in the **Network Info** window. Select an element such as the **Nodes** tab, toggle the up and down arrow to view more details, and click the **User Parameters** tab. This can be done for **Links**, **Sites**, **Demands**, and **Tunnels** as well.

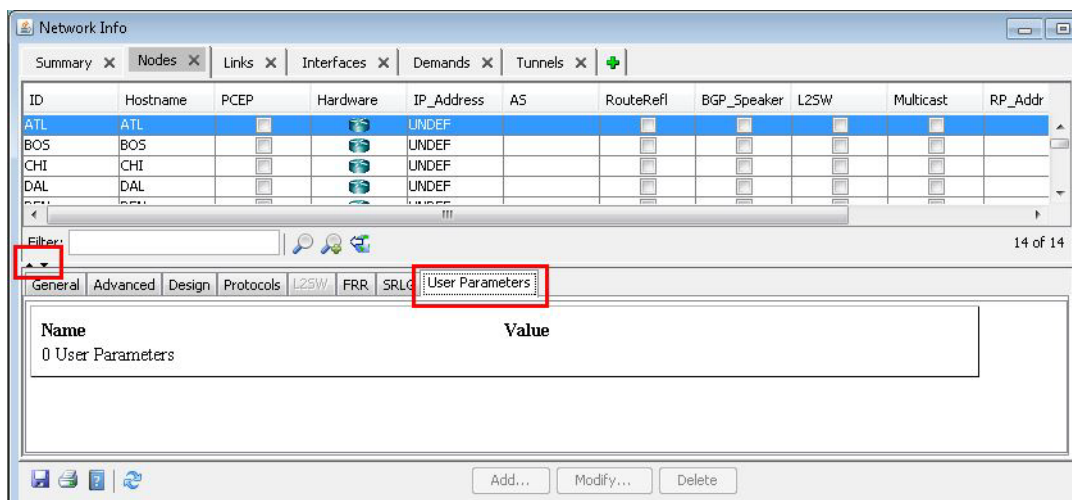


Figure 75: Nodes Tab User Parameters

1. In the **Nodes** tab, select a node entry, and click **Modify**.
2. Go to the **User Parameters** tab.
3. Select a User Parameter entry and type in a value for it. Click **OK** to make changes. This can also be done for **Links**, **Sites**, **Demands**, and **Tunnels**.
4. When modifying more than one element at a time, a dialog box will provide the option to append or overwrite onto any existing value.

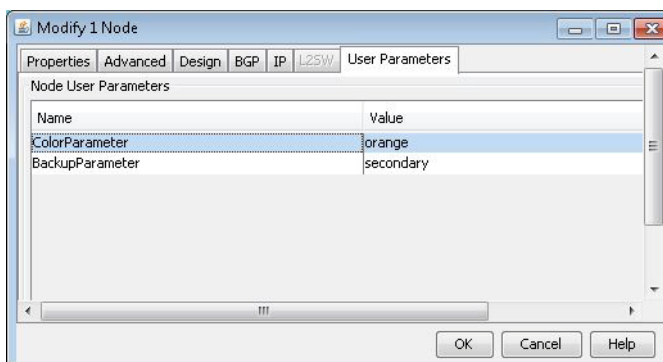


Figure 76: Modify User Parameter for a Node

Performing Text Filter on User Parameters

One useful feature of the User Parameters feature is the ability to use them as custom keys when searching with filters on **Nodes**, **Links**, **Demands**, or **Tunnels**.

1. In the **Network Info** window **Nodes** tab, click the **Advanced Search** icon.

2. The User Parameter entries will appear in the Keys list. Insert the items for your query and click **OK** to perform the filter.

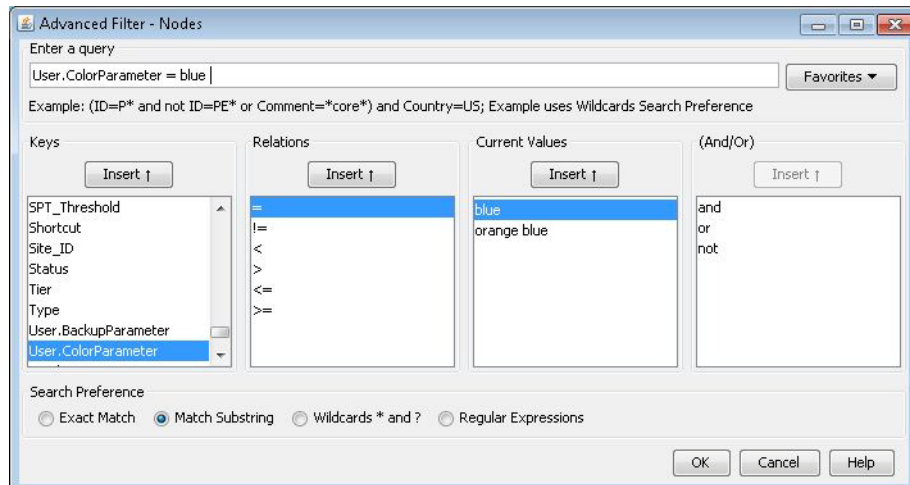


Figure 77: Advanced Search

Map Labels with User Parameters

User Parameters can be labeled on the Map. Right-click on the Map and select **Labels > Node Labels** or **Link Labels > Customize**. The User Parameter keys are listed together in one group called “User” at the top of the list of keys. The

The Usrparam File Format

User Parameters are saved in the **usrparam** file. Following is an example usrparam file:

```
Node,,city state
NODE,CHI,city=Chicago;state=Illinois;
NODE,DEN,city=Denver;state=Colorado;
LINK,,description
LINK,LINK4,description=my_description;
TUNNEL,,mylabel
TUNNEL,ATL,RATLCHI,mylabel=test;
```

There are two Node parameters: city and state. The subsequent 2 lines define node CHI's city as Chicago and state as Illinois, and node DEN's city as Denver and state as Colorado.

The usrparam file consists of 2 types of lines-- one to identify the attribute names, and the other to provide the attribute/value pairs.

The first keyword defines the ElementType: NODE, LINK, or TUNNEL.

- If this keyword is followed by 2 commas, the attribute names are provided.
- If this keyword is followed by 1 comma, the attribute/value pairs are provided for the indicated element.

Attribute Declarations

```
#ElementType,,[list of attribute names separated by spaces...]
```

Attribute/Value Pairs for Nodes and Links

```
#ElementType,ElementName,[Attribute=Value;]*
```


Attribute/Value Pairs for Tunnels

```
#ElementType,HeadEnd,ElementName,[Attribute=Value;]*
```

Note that in the case of tunnels, the element has to be identified not only by the tunnel name but also by the source device, since a tunnel name is only required to be unique for its host and not network-wide. In the example above, the tunnel whose attribute/value pair is defined has its headend in node ATL, and has the tunnel name RATLCHI.

Modifying Link Pricing

This chapter describes how to use the pricing features of IP/MPLSView. The user can take advantage of a standard database or specify pricing rules in accordance with the IP/MPLSView file formats.

When to use

In IP/MPLSView, pricing information is taken into account when performing least-cost network designs. Use these procedures to understand how pricing works and to price links.

Prerequisites

You should have started up IP/MPLSView and opened a specification file as described in Chapter 4, Opening and Closing a Network Project.

Related Documentation

You may also want to refer to the IP/MPLSView Java-Based Graphical User Interface Reference chapter on the Network menu.

Recommended Instructions

Following is a high-level sequential outline of the process of viewing network information and recommended detailed procedures.

For some of the example, this tutorial uses the sample network specification file `/u/wandl/sample/original/router/wandl/spec.wandl`. If you do not have permission to use this example, you may edit the hardware vendor listed in the `dparam` file as "hwvendor=ROUTER" to a hardware that you do have.

To do this, double-click on the `dparam` file in the **File Manager** and find and edit this line. Note that this may affect the routing and therefore the results you get may be different than that obtained for the router hardware.

- View the current price of the network links.
- Create a new `custrate` file through the JAVA interface.

Detailed Procedures

Custom Rates

The custom rates feature allows you to define, or customize, your own link rate table.



Informational Note: The custom rates feature requires a license. For more information, please contact your Juniper representative.

The basic idea is to first categorize your nodes into certain classes, called custom rate classes. Then, define the link pricing formula for various trunk types that are purchased between nodes of particular classes. Following is a list of some possible applications for customized link rates:

- You wish to model different pricing schemes for links between different countries or regions. In this case, the custom rate classes can be used to classify nodes based upon country or region.
- Your network defines the hierarchy level of the nodes (for example CORE, REGULAR, ACCESS, and so on.) and you would like to encourage/discourage the purchase of links between certain hierarchy level types. Custom rates can be used to influence the network design, because IP/MPLSView will compute a design that minimizes the network cost.
- You have your own method of categorizing nodes, on which the pricing schemes can be based.

Using the custom rates feature, the final recurring charge (for example, a monthly charge) for a link is calculated according to the following formula:

- $\text{Recurring_charge} = \text{Fixed_cost} + (\text{Inc_cost} / \text{Inc_dist}) * \text{link_airline_distance}$

where the value of Inc_cost and Fixed_cost (the incremental and fixed costs, respectively) are taken from the distance band that applies for the link.

1. From the main menu, select **Network > More > Custom Rates > View Classes**.

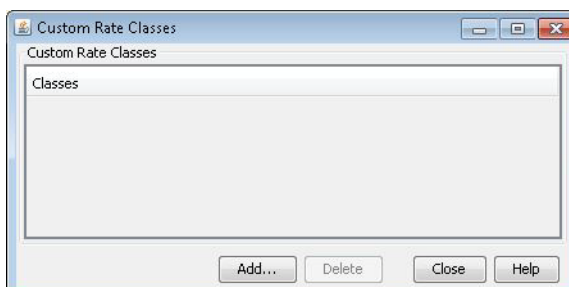


Figure 78: Custom Rate Classes

2. Click **Add** button and enter a name for this class of node and then click **OK**. For this sample, enter the name "US" to represent nodes in the United States, with the intent of using different pricing schemes for links between different countries or regions.

3. To apply this custom rate class, open the **Network Info** window **Nodes** tab. Filter or select the nodes you wish to classify as “US” and click **Modify**. In the **Advanced Properties** tab, change the Custom Rate Class field to “US” and click **OK**.

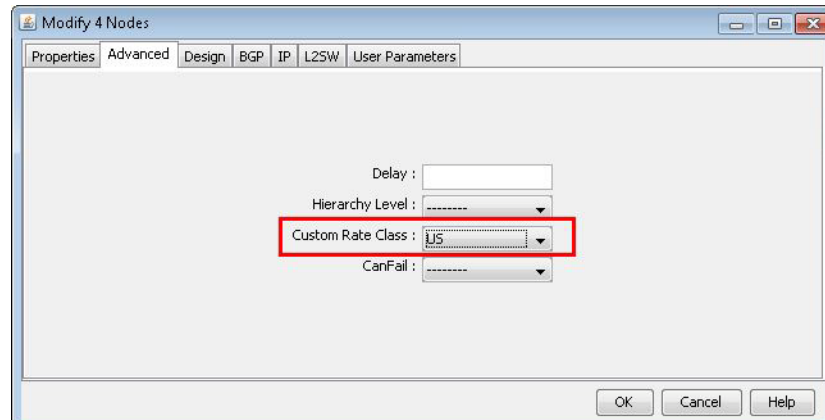


Figure 79: Applying Custom Rate Class for Nodes

4. Open the **Custom Rate Class** window again, click **Add** to enter a second class name. For example, “Intl”. Apply this custom rate class (“Intl”) to a different set of nodes.
5. From the main menu, select **Network > More > Custom Rates > View Bands**.

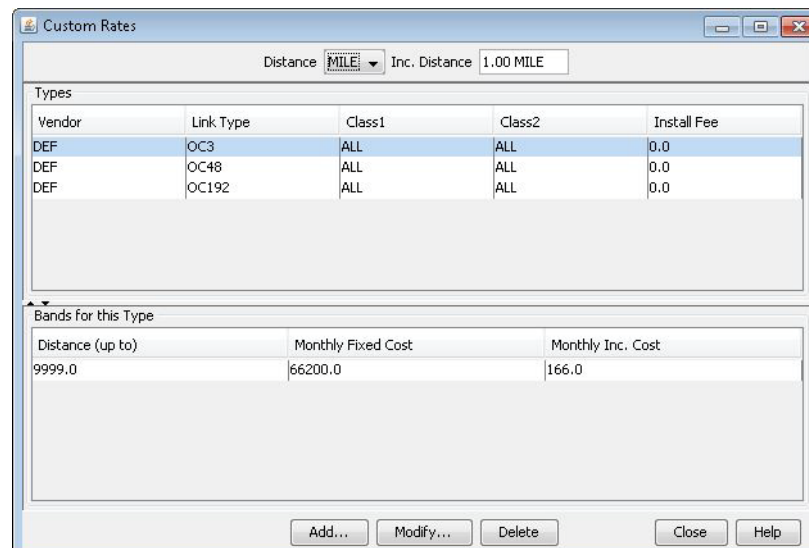


Figure 80: Custom Rates

6. Click **Add** to add a new custom rate entry. In the **Link Type** field, select a trunk type in your network (for example, ET100M) by selecting it from the drop-down menu or typing it in. Next, select the class “US” for both **Class1** and **Class2**. This means that this custom rate will be applied to links of the specified trunk type with one end node in Class1 and one end node in Class2. The class “ALL” refers to the set of all nodes.
7. The **Install Fee** is a non-recurring charge. Enter a number (for example, 1000).

8. You can enter in distance bands and the recurring fixed cost and incremental cost that applies to each link whose length falls within that distance band. The “monthly fixed cost” refers to a recurring cost that is independent of the link length. The “monthly inc. cost” refers to a recurring cost that is dependent on the link length and that applies for each incremental distance.

The first distance band starts from 0 to the first number entered under **Distance (upto)**. The second distance band starts from the first Distance number to the second Distance number, and so forth. The last distance value should be 9999 or higher.



Informational Note: If the corresponding distance band does not automatically display, select an entry of the service type to display it.

9. You can add additional rows by clicking on the **Add Row** button on the right side of the window.
10. When you are finished, click **OK**. Then click **Close** in the **Custom Rates** window..



Informational Note: If the price remains 0*, there are two possible reasons why. It is possible that there are no links falling under the user-specified link service types. Alternatively, the length of certain links (airline mileage between source and destination node) may be unknown. The geographical location of the link’s end nodes should be specified. In the US, it is sufficient to specify the “npa nxx.” Otherwise, one can specify the latitude and longitude to allow the program to solve for the geographical (airline) distance.

Distance (up to)	Monthly Fixed Cost	Monthly Inc. Cost
1000.0	100.0	15.0
2500.0	200.0	25.0
9999.0	0.0	0.0

Figure 81: Modify Custom Rates

Chapter 14

Design

This chapter describes how to resize your links based on the traffic utilization for normal or failure conditions, and how to create a simplified network model.

When to use

Use the Resize feature to adjust the bandwidth of the backbone links based on the normal traffic load, or based on peak simulation results.

Use the Create Simplified Group Model feature to consolidate nodes and its connecting links and demands.

Prerequisites

You should know how to add and modify your network, as explained in Chapter 9, Adding Nodes, Links, and Demands and Chapter 10, Modifying Nodes, Links and Demands.

Related Documentation

For a detailed description of the design options, refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Refer to the File Format Reference for IP/MPLSView for a detailed description of the file formats.

Recommended Instructions

Following is a high-level sequential outline of the recommended procedures.

- The Resize Feature on page 103.
- Simplified Group Model Walk-Through on page 106.

Detailed Procedures

The Resize Feature

The Resize feature allows you to resize links based on normal traffic utilization or based on peak traffic utilization from failure simulation.

Using the sample network in /u/wandl/sample/IP/fish/spec.mpls-fish, this exercise will grow the traffic tenfold, and then run through the exercise of resizing the links accordingly.

1. Open the sample network and modify the demands via in the **Network Info** window **Demands** tab.

2. Click **Modify > All Entries**, and check **Multiply BW by ___%**, entering the value 1000 to increase the bandwidth of each demand 1000% or tenfold.
3. Click **OK** to apply the changes.
4. From the main menu, select **Design > Design Options > Resize** tab.

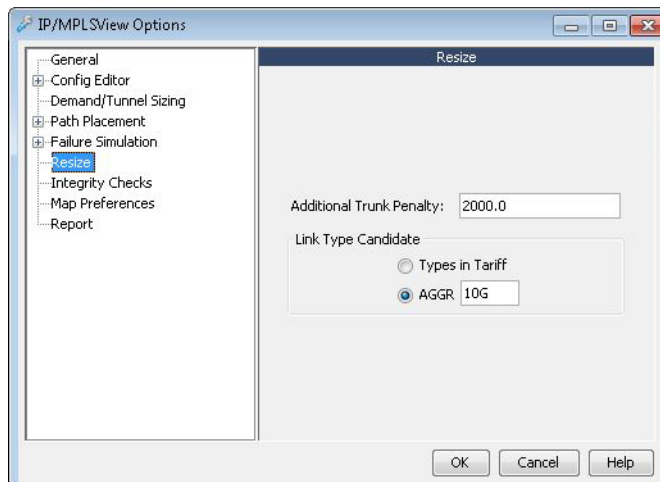


Figure 82: Resize Options

5. Enter the **Link Type Candidates** to be used for the Resize. For the aggregated link AGGR, select the bandwidth increment that can be used for the design. With the default of 10G, links will be resized to increments of 10G: AGGR10G, AGGR20G, AGGR30G, and so on.
6. For this exercise, change the AGGR bandwidth to 1G, so the links can be resized to increments of 1G: AGGR1G, AGGR2G, AGGR3G, and so on.
7. The **Additional Trunk Penalty** field allows for a penalty to be set for the purchase of each additional trunk so as to discourage the replacement of a high capacity trunk with several smaller capacity trunks during resize. The default value is 2000.
8. Click **OK** to apply the changes.
9. Next, from the main menu, select **Design > Backbone > Resize** to see the following available menu options.

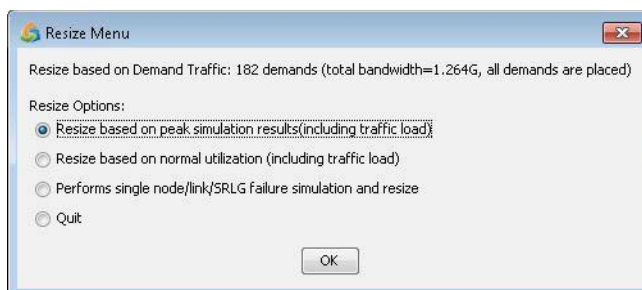


Figure 83: Resize Prompt

10. Select **“Resize based on normal utilization”** to resize based on the trafficload file only.

11. If you have performed failure simulation before starting the **Resize** feature, select **"Resize based on peak simulation results"** to resize the links based on those peak utilization results.
12. Select **"Perform single node/link/SRLG failure simulation and resize"** to automatically run exhaustive failure simulation for nodes, links, and SRLGs, and then resize based on those peak utilization results.
13. For this exercise, select **"Resize based on normal utilization"**.
14. The subsequent prompt allows you to take all recommended updates, either for all links, or only overloaded links. Or you can view and manually select which links to update.

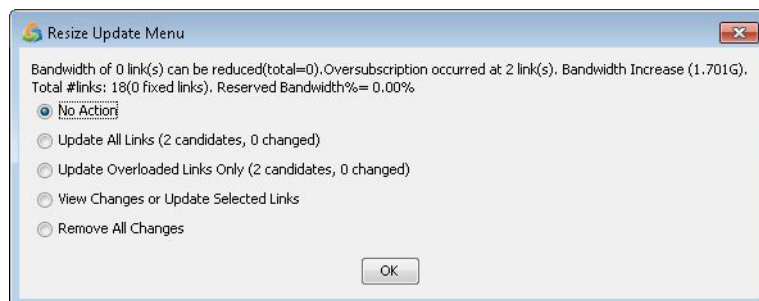


Figure 84: Resize Update Prompt

15. Select **"View Changes or Update selected Links"**.

Resize	LinkName	NodeA	NodeZ	Type	LinkSpe...	UsedB...	UtilPct	Sugges...	Sugges...	Saving(...)	NewType	NewCo...	Hardw...	Hardwa...
<input type="checkbox"/>	LINK1	ATL	HOU	OC3	155.520M	24.082M	15.48%		15.48%				CISCO	JUNIPER
<input type="checkbox"/>	LINK18	ATL	LAX	OC3	155.520M	144.240M	92.75%		92.75%				CISCO	JUNIPER
<input checked="" type="checkbox"/>	LINK2	ATL	WDC	OC3	155.520M	173.584M	111.62%	AGGR1G	17.36%	-850.24M	AGGR2	1	CISCO	BROCADE
<input type="checkbox"/>	LINK3	BOS	DET	OC3	155.520M	79.417M	51.07%		51.07%				CISCO	JUNIPER
<input type="checkbox"/>	LINK4	BOS	NYC	OC3	155.520M	35.683M	22.94%		22.94%				CISCO	JUNIPER
<input type="checkbox"/>	LINK5	CHI	DAL	OC3	155.520M	105.868M	68.07%		68.07%				CISCO	CISCO
<input type="checkbox"/>	LINK6	CHI	DEN	OC3	155.520M	68.270M	43.90%		43.90%				CISCO	CISCO
<input type="checkbox"/>	LINK7	CHI	DET	OC3	155.520M	101.079M	64.99%		64.99%				CISCO	JUNIPER
<input type="checkbox"/>	LINK8	CHI	WDC	OC3	155.520M	130.676M	84.03%		84.03%				CISCO	BROCADE
<input type="checkbox"/>	LINK9	DAL	HOU	OC3	155.520M	97.729M	62.84%		62.84%				CISCO	JUNIPER
<input type="checkbox"/>	LINK10	DEN	SFO	OC3	155.520M	38.952M	25.05%		25.05%				CISCO	BROCADE
<input type="checkbox"/>	LINK11	HOU	SDG	OC3	155.520M	61.818M	39.75%		39.75%				JUNIPER	BROCADE
<input type="checkbox"/>	LINK12	LAX	SDG	OC3	155.520M	86.112M	55.37%		55.37%				JUNIPER	BROCADE
<input type="checkbox"/>	LINK13	LAX	SJC	OC3	155.520M	172.688M	111.04%	AGGR1G	17.27%	-850.24M	AGGR1G	1	JUNIPER	BROCADE
<input type="checkbox"/>	LINK14	NYC	PHI	OC3	155.520M	149.686M	96.25%		96.25%				JUNIPER	JUNIPER
<input type="checkbox"/>	LINK15A	PHI	WDC	OC3	155.520M	70.659M	45.43%		45.43%				JUNIPER	BROCADE
<input type="checkbox"/>	LINK15B	PHI	WDC	OC3	155.520M	95.577M	61.46%		61.46%				JUNIPER	BROCADE
<input type="checkbox"/>	LINK16	SFO	SJC	OC3	155.520M	122.262M	78.62%		78.62%				BROCADE	BROCADE

Figure 85: Updating Selected Links

This option allows you to select which links to update, and to modify the resulting trunk type or count as desired. Once the checkbox is marked, the NewType and NewCount columns will become modifiable for that row. For example, the user could change AGGR1G to AGGR2G. If manually entering a new bandwidth, do not forget to enter the unit of measure.

16. Select **Resize Selected Links** to update the rows which have been updated, or Cancel, to return to the previous menu.
17. When all the desired changes have been made, select **Quit**.

Create Simplified Group Model

The Create Simplified Group Model helps simplify the network topology by consolidating multiple nodes at the same site into one node. The consolidated node keeps the inter-site traffic however intra-site traffic is ignored. This means the links and demands between sites are kept for modeling while links and demands within the site are discarded. This creates a simplified network topology where sites are represented as single nodes and the traffic matrix is re-mapped to this simplified network topology.

Simplified Group Model Walk-Through

1. Open the spec.mpls-fsh network located at /u/wandl/sample/original/router/fish/
2. Select and Group nodes BOS, NYC, and PHI into a group called EAST.
3. Select and Group nodes SFO, SJC, LAX, and SDG into a group called WEST.
4. Select and Group the remaining nodes into a group called SUPERCORE or SUPERGROUP. Both are special keywords that can be used for grouping. Nodes in this special group will not be consolidated.

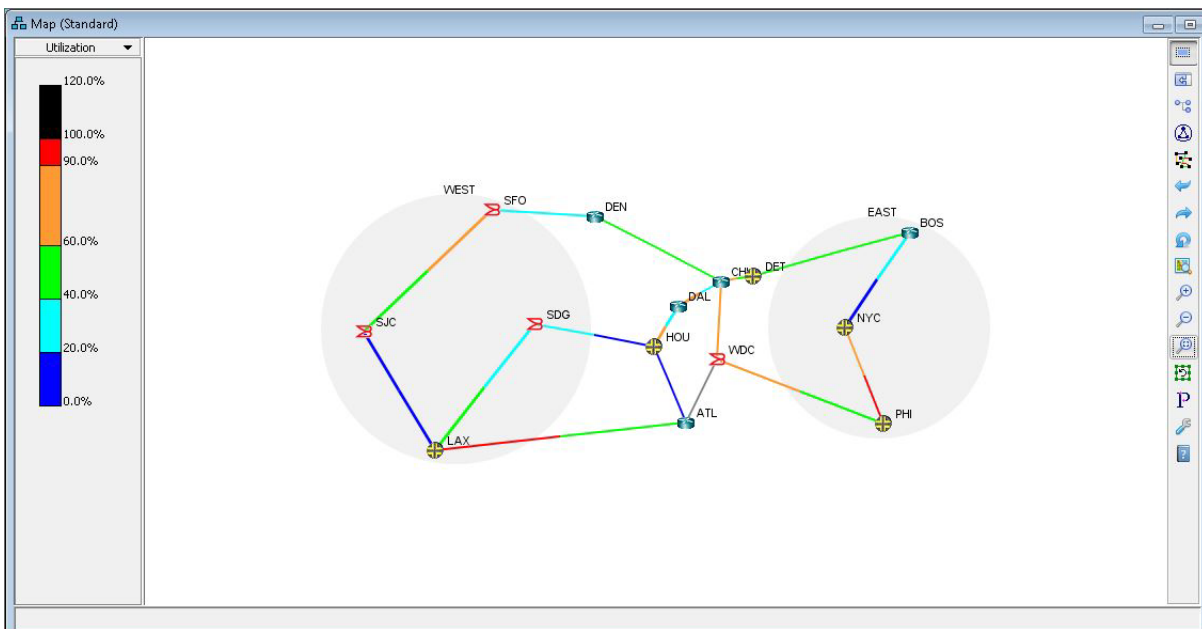


Figure 86: Grouping nodes

- Open Report Manager and select **Network Reports > Group > Group Demand Summary by Group Pair**. The report displays the demand BW between group (inter) and among the group (intra). Keep note that the total demand BW=1.264G; with intra EAST=78.212MB and intra WEST=181.927MB.

Calculation Method: Analytical

Group Type: User Defined Group Period: All Generate

Explanation... Multiple Sort... Restore Select Columns... Re-Generate Script... P_: All

Group A	Group Z	Intra/Inter	Max. Delay	Min. Delay	Avg. Delay	#Demand	BW(Mbps)
-	-	INTRA	26.30	2.35	12.37	42	300.301
EAST	EAST	INTRA	2.68	0.78	1.78	6	78.212
WEST	WEST	INTRA	4.56	0.51	2.77	12	181.927
-	EAST	INTER	30.50	1.33	12.96	21	149.186
-	WEST	INTER	28.20	9.49	19.68	28	97.747
EAST	-	INTER	26.78	1.33	12.08	21	174.416
EAST	WEST	INTER	42.60	26.10	32.59	12	40.302
WEST	-	INTER	29.67	9.49	20.27	28	152.835
WEST	EAST	INTER	42.60	26.10	32.59	12	89.229
Delay Sum...	3	INTRA	11.18	1.21	5.64	60	
Delay Sum...	6	INTER	33.39	12.31	21.69	122	
Delay Sum...	9	BOTH	25.99	8.61	16.34	182	

Filter: * 12 of 12 displayed

Figure 87: Report Manager Groups

- From the main menu, select **Design > Backbone > Create Simplified Group Model**.
- The Info pop-up window describes the actions that will happen:
 - A new network model will be saved using runcode "mergegroup".
 - A new node will be created to replace each group.
 - Demands, tunnels, and links terminating at nodes in a group are re-terminated to that new node.
 - Nodes not in a group are left alone.
- Click Save simplified network in current directory with runcode=mergegroup.

9. Open the newly created spec.mergegroup.

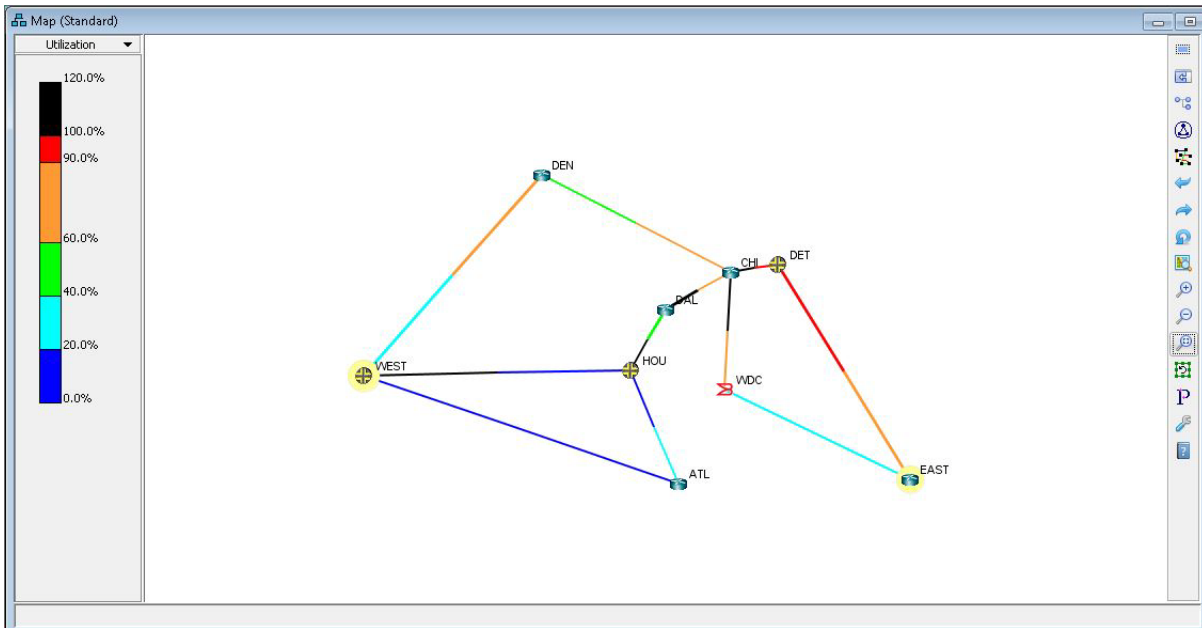


Figure 88: Simplified Network Topology

All the nodes in the EAST and WEST groups no longer exist and are consolidated simply as a single node named EAST and WEST.

10. Go to menu **Network > Summary**. The total demand BW is now 1.004G. This is now different than the original 1.264G because the intra EAST and WEST demands were discarded during consolidation. In Step 5 total traffic before was 1.264G, minus intra EAST 78.212MB, minus intra WEST 181.927MB = 1.004G.

Network Info			
Summary x Nodes x Links x Interfaces x Demands x Tunnels x +			
Demand Placement			
		Demands	Bandwidth
Placed	164		1.004G
Unplaced	0		0
Deactivated	0		0
Total	164		0

Figure 89: Network Summary of Mergegroup

11. **Optional:** To further verify traffic between sites, you can assign the EAST, WEST, and other remaining nodes into the 3 separate groups and check the Group Summary report in Report Manager.

This exercise demonstrates the functionality of Create Simplified Group Model. It can help consolidate the network topology and simply design, plan, and what-if studies.

Simulation

This chapter describes how to perform simulations in your network.

In Failure Analysis, path provisioning simulates the hardware's implementation of bandwidth allocation and demand routing on the existing topology. You can run interactive simulations or simulation scripts (predefined or user-created). In addition, you can try simulating path placement for several iterations to see what impact demand path placement and demand routing order have on the network.

Backbone Simulation

The Backbone Simulation module performs dynamic routing simulations of demands in the backbone network. It is assumed that all paths are initially placed by the hardware at set-up. Once placed, a demand is rerouted only when network failure occurs or when it is preempted by a higher priority demand.

When a node or link failure occurs, the simulation module attempts to find new routes for all of the disconnected demands. Higher-priority demands are placed before those with lower-priority. If the network does not have enough bandwidth for a demand, the simulation will take bandwidth from a minimum set of demands whose priorities are less than the current demand. The simulator then attempts to find new routes for the demands it has disconnected. If more than one choice is available, the path with the shortest distance is selected. To simulate unpredictable timing effects, disconnected demands with the same priority are placed in varying order in simulation runs. If a demand has a preassigned path associated with it, the simulation always tries to use the preassigned path before searching for other paths.

Network nodes and links can be brought up or down sequentially to study the impact on path placements. Information such as link utilization, demand failures, and network failures can be accessed using the interactive graphics interface.

When to use

Simulations may be performed on both new and existing designs to determine resiliency in the backbone. Many hypothetical "What If?" situations can be simulated using network failure simulation. Examples include single line failure, link failure, node failure, multiple element failure (random daily failure), and a user-defined sequence of events.

Using the random daily failure simulation, the program can randomly bring down a given number of links daily, for a set number of days. Extreme failure scenarios such as the outage of an entire carrier at a given number of sites can be studied. In the interactive mode, you can activate and deactivate nodes and links in any order and study the impact of the failure. IP/MPLSView will display which circuits are disconnected, and the new paths taken if the circuits are rerouted successfully. It is also possible to look for worst case scenarios by placing low bandwidth demands first.

To study the impact of preempt priority, the “bumpflag” parameter may be used. If bumpflag is set to 0, lower-priority paths are NOT bumped by higher-priority paths. If this parameter is set, the preempt value specified in the demand file is not considered. This allows you to study the impact of the bump feature.



Informational Note: The reserved bandwidth specified by the “fat” parameter and the reserved bandwidth file are both ignored by the simulation module (the reserved bandwidth is only used for design purposes). The “hopdelay” parameter is also ignored.

Prerequisites

Before beginning this task, you must have created a specification file as described in Chapter 5, Creating and Editing a Network Model or opened a specification file as described in Chapter 4, Opening and Closing a Network Project. You then should continue with these procedures to perform simulations.

To follow along with this example, you should use the specification file spec.dog in \$WANDL_HOME/sample/original/router/dog. If you do not have the appropriate license to open this file, see Following Along With the Examples in This Manual on page 1. Note that results may vary depending upon the device hardware.

Related Documentation

For a detailed description of the Simulation windows, refer to the simulation menu and failure simulation options topics in the IP/MPLSView Java-Based Graphical User Interface Reference.

Recommended Instructions

Following is a high-level sequential outline of the simulation process and recommended procedures.

1. Perform interactive simulations (fail links, nodes, and facilities) as described in Interactive Simulation on page 111.
2. Try a failure simulation script (exhaustive single or double node failure and link failure) and survey the script options as described in Simulation Scenarios on page 117.
3. Analyze failure simulation reports as described in Simulation Reports on page 118.
4. View simulation results graphically on the map as described in Viewing Simulation Results Graphically on page 121.

Detailed Procedures

Following are the detailed procedures for the simulation process, as well explanations of the steps involved.

1. Open the spec.dog network.
2. From the main menu, select **Tools > Options > Failure Simulation**. These options determine what to include or exclude when running simulations.

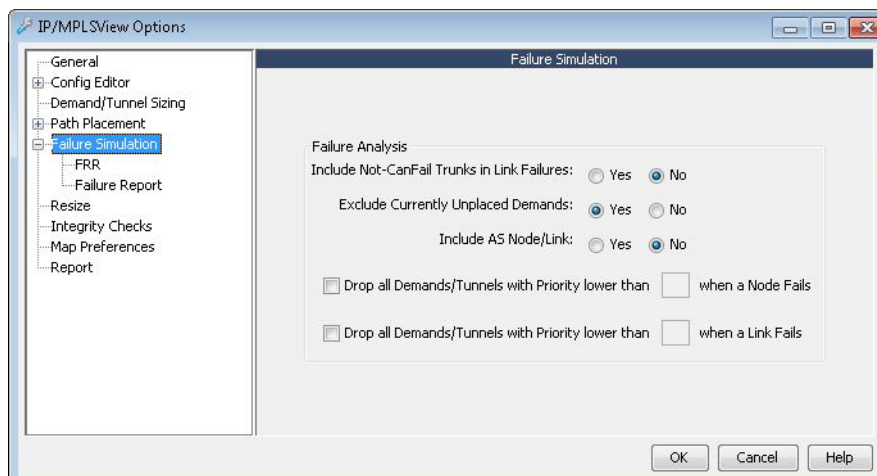


Figure 90: Failure Simulation Options

Interactive Simulation

Interactive Simulation allows you to indicate the specific nodes, links, and SRLG's (facilities) to be failed during a simulation run.

During the simulation, all the demands that are disconnected due to the specified failures are rerouted one at a time, according to the path placement options specified in the Failure Simulation Options.

Interactive Link Failure

1. From the main menu, select **Simulation > Interactive Scenarios** and then click the **Link** tab. Each row contains information for a link.

- This simulation option allows you to select links to be failed in the network. To fail an element, click on an entry and the **Fail** column will indicate which elements are down during failure simulation. To bring an element back up, re-click on it. The **Reset Simulation** button will globally bring all failed elements up.

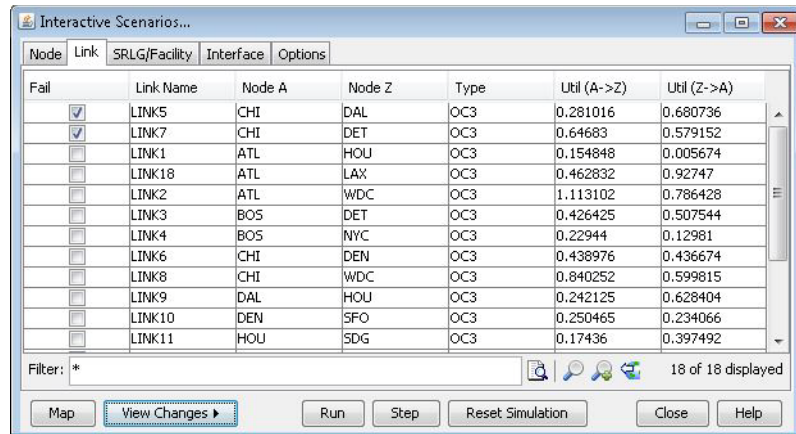


Figure 91: Interactive Fail Links window

- When elements are selected for failure, the console will report that those elements are down. For example, here is the resulting console message from failing LINK1 and LINK3

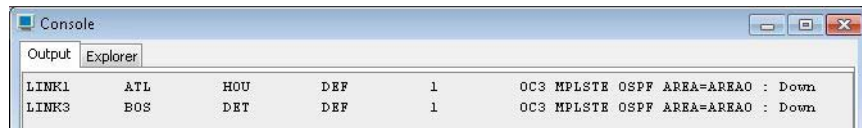


Figure 92: Console Link Down Messages

Alternatively, you can fail a link directly from the map. This requires having the **Interactive Scenarios** window open. Select one or more links on the map to highlight the links to fail. Then right-click over one of the selected link(s) and select **Fail Selected Links**.

- Right-click on the console and turn the **Trace** option on and then select a trace file name in which to save the simulation output. You can stop the trace by right-clicking on the console and selecting the toggling **Trace** option once again to turn it off.
- To start the simulation, click on the **Run** button. At the end of the simulation, there will be some console messages. The links brought down during the simulation will be colored gray and marked with a red 'F' symbol on the Map. From the trace file or from the console, you will see summary statistics before and after the failure simulation. These include the max hop count and average hop count for demands, and information on failed demands including the count, bandwidth of failed demands. The final column indicates how many links were oversubscribed (traffic exceeded link bandwidth) before and after the failure simulation.

For this spec.dog router network, the report also indicates links that experienced a “**Reserved Bandwidth Violation**” during the simulation. That is, after the demands are rerouted, certain links are over 100% utilized. This flags potential congestion and dropping that may occur in the event of particular failures.



Informational Note: If you are running failure simulations in an ATM network, you should not encounter this scenario. If there is insufficient bandwidth, the demand will fail to be rerouted, and its status will be reported as “**FAILED**”.

6. Click the **Reset Simulation** button to restore all devices and reset the simulation to the initial network state. All links, nodes, and facilities are brought back up, and demands are routed according to the state prior to simulation.

Interactive Node Failure

Try failing node ATL and node BOS and then run a simulation. Select the **Node** tab from **Simulation > Interactive Scenarios**.

When you click on the **Run** button, you see simulation results in the **Console**, followed by a listing of disconnected demands and how they are handled, followed by a summary section.



Informational Note: Once the **Interactive Scenarios** window is open, do not press the **Close** button until your simulation has completed. Pressing the **Close** button will cause the failed elements to be brought back up.

Interactive SRLG/Facility Failure

An SRLG, or shared risk link group, is a group of links likely to fail together in the event of a failure.

A facility is a group of links and/or nodes that are likely to fail together in the event of a failure. This option can be used in failure simulations where one wants to fail elements in such a group simultaneously. One example is to group a set of links that go through the same fiber into a facility so that when the fiber is cut, the links in that facility are failed together. Alternatively, one may want to group together the links that go through a particular card.

1. To create a facility, select **Network Info > Elements > SRLGs** to open the **Network Info** window **SRLGs** tab. Click **Add**, select both LINK15A and LINK15B from the link table on the right, enter a facility name (for example, "FACILITY15"), and click **OK**.

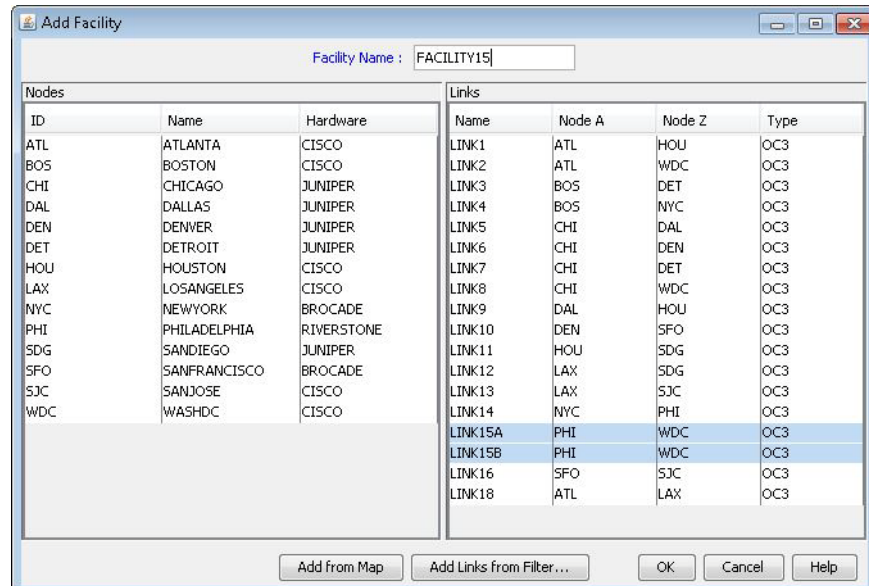


Figure 93: Add Facility Window

2. Open the **Simulation > Interactive Scenarios > SRLG/Facility** tab, and fail "FACILITY15".

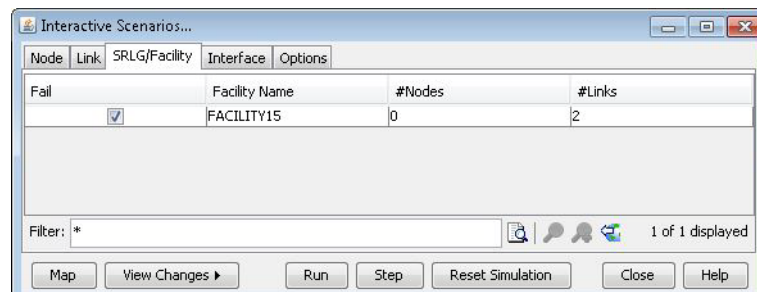


Figure 94: Interactive Scenarios Window SRLG Tab

3. Click **Run** to start the failure simulation.
4. Observe the results in the **Console**.
5. Click **Reset Simulation** to start over.

Stepping Through a Simulation

For a more detailed failure simulation analysis, rerun the simulation. This time set **Verbose Mode** in the **Options > Failure Simulation > Failure Report** tab and use **Step** instead of **Run**. Select the links to fail and then click **Step**. Stepping will allow you to see how each disconnected demand is rerouted one by one.

For each disconnected demand there are two lines in the console. One is a line indicating the original path and ending with “DISCONNECTED.” Following that is another line for the demand either indicating a new path followed by “REROUTED” or the old path followed by “FAILED” if that demand was not able to be routed. The following is an example of the console after a link failure.

```

Console
Output Explorer
LINK7 CHI DET DEF 1 OC3 OSPF AREA=AREA0 : Down
--- 182 demands don't have traffic load info
Traffic load for these demands are initialized according to its bandwidth, util
* Simulation Options Used:
* Algorithm: Shortest Path Algorithm
* Link distance = Admin Cost
* randomflag = hardware default,* Path is selected randomly among paths
of the same length during path selection
* Max Hop Allowed= 20,
* Path placement order= High priority, high bandwidth demands first, scramble

Start Simulation:
* * *
All 182 demands placed before configuration changes
22 more demand(s) are disconnected due to new failures
xflow14 CHI BOS 22.2919M R,A2Z 02,02 CHI--DET--BOS #!delay=8.467ms DISCONNECTED
xflow14 CHI BOS 22.2919M R,A2Z 02,02 CHI--WDC--PHI--NYC--BOS #!delay=9.84ms REROUTED
xflow16 DEN BOS 714K R,A2Z 02,02 DEN--CHI--DET--BOS #!delay=17.661ms DISCONNECTED
xflow16 DEN BOS 714K R,A2Z 02,02 DEN--CHI--WDC--PHI--NYC--BOS #!delay=19.034ms REROUTED
xflow15 DAL BOS 651.231K R,A2Z 02,02 DAL--CHI--DET--BOS #!delay=16.523ms DISCONNECTED
  
```

Figure 95: Console Link Failure Messages

A **Paths** window will open displaying a graphical view of the demand routing before and after the demand was disconnected.

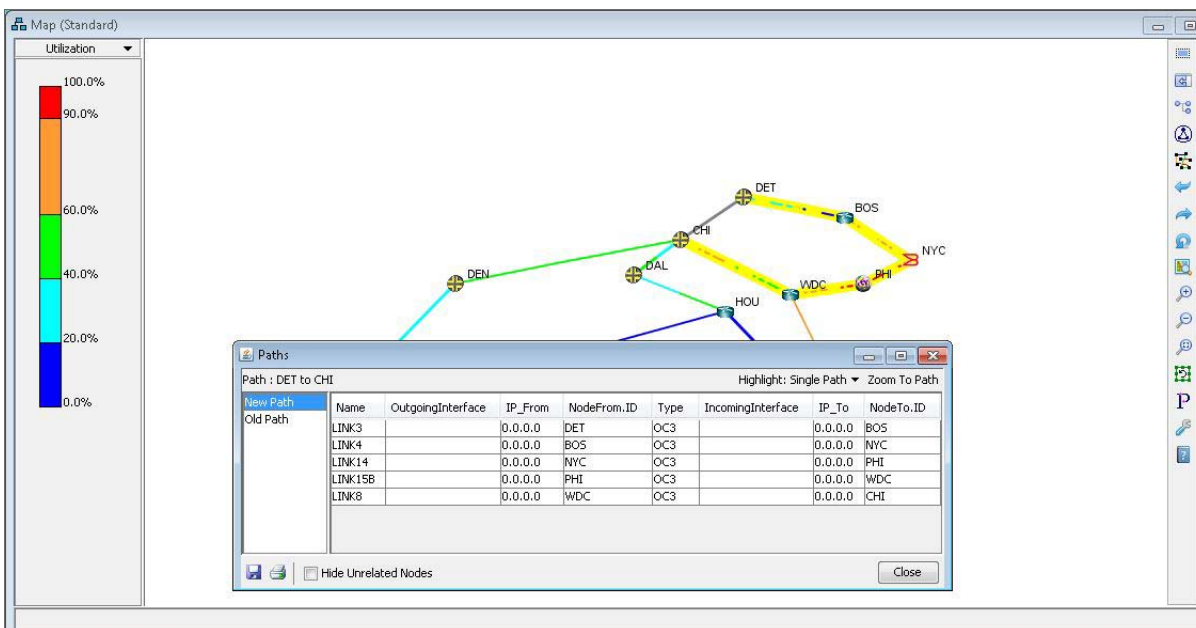


Figure 96: Paths Window

Continue to **Step** through more disconnected demands, or click **Run** to simulate all demands. You can stop the simulation by clicking the **Stop** button.

When you are done with the failure simulation, click **Reset Simulation** to restore all devices.

View Changes

1. After the simulation finishes, select **View Changes > View Link Changes** to see the impact of the simulation upon the link utilization and RSVP utilization.
2. Select **View Changes > View Demand Changes** to see the impact of the simulation upon the demand routing, delay, and path cost.
3. Select **View Changes > View Tunnel Changes** to see the impact of the simulation upon the tunnel routing, delay, and path cost.

LinkName	Node:Inter...	RemoteNode	LinkBwCha...	LinkBw	NewUtil	OrigUtil	UtilDiff	New_RSVP...	Orig_RSVP...	RSVP_UtilDiff
LINK1	ATL	HOU		155.52M	0.4609	0.1548	0.3061	0.0000	0.0064	-0.0064
LINK1	HOU	ATL		155.52M	0.4878	0.0057	0.4821	0.0000	0.0000	0.0000
LINK2	ATL	WDC		155.52M	1.5792	1.1131	0.4661	0.0000	0.0000	0.0000
LINK2	WDC	ATL		155.52M	0.9172	0.7864	0.1307	0.0000	0.0000	0.0000
LINK3	DET	BOS	link down	155.52M	-down-	0.5075	--	--	0.0965	--
LINK3	BOS	DET	link down	155.52M	-down-	0.4264	--	--	0.0643	--
LINK4	BOS	NYC		155.52M	0.5637	0.2294	0.3342	0.0000	0.0000	0.0000
LINK4	NYC	BOS		155.52M	0.5452	0.1298	0.4154	0.0000	0.0000	0.0000
LINK5	DAL	CHI	link down	155.52M	-down-	0.6807	--	--	0.0707	--
LINK5	CHI	DAL	link down	155.52M	-down-	0.2810	--	--	0.0000	--
LINK6	CHI	DEN		155.52M	0.5893	0.4390	0.1503	0.0000	0.0000	0.0000
LINK6	DEN	CHI		155.52M	0.6481	0.4367	0.2114	0.0000	0.0000	0.0000

Figure 97: Changed Links

Name	Node A	Node Z	New Path	Orig Path	New Path ...	Orig Path ...	New # Hops	Orig # Hops	New Delay	Orig Delay
flow1	ATLANTA	BOSTON	ATL--WDC--...	ATL--WDC[...	1442	4354	4	4	9.36	19.77
flow2	ATLANTA	CHICAGO	ATL--WDC--...	ATL[(T=RAT...	2481	3126	2	3	11.24	17.24
flow4	ATLANTA	DENVER	ATL--WDC--...	ATL[(T=RAT...	4694	5339	3	4	20.30	26.30
flow5	ATLANTA	DETROIT	ATL--WDC--...	ATL--WDC[...	2958	5750	3	5	13.64	25.90
flow8	ATLANTA	NEWYORK	ATL--WDC--...	ATL--WDC--...	1060	1060	3	3	7.47	7.47
flow14	BOSTON	CHICAGO	BOS--NYC--...	BOS--DET--...	2557	1873	4	2	9.88	8.53
flow15	BOSTON	DALLAS	BOS--NYC--...	BOS--DET--...	3313	3128	6	3	18.73	16.40
flow16	BOSTON	DENVER	BOS--NYC--...	BOS--DET--...	4770	4086	5	3	18.94	17.59
flow17	BOSTON	DETROIT	BOS--NYC--...	BOS--DET--...	3034	1396	5	1	12.28	6.13
flow18	BOSTON	HOUSTON	BOS--NYC--...	BOS[(T=RB...	2880	5792	5	5	16.38	26.78
flow19	BOSTON	LOSANGELES	BOS--NYC--...	BOS[(T=RB...	4442	7354	5	5	28.77	39.18
flow22	BOSTON	SANDIEGO	BOS--NYC--...	BOS[(T=RB...	4705	7617	6	6	29.91	40.32
flow23	BOSTON	SANFRANCI...	BOS--NYC--...	BOS[(T=RB...	5222	8134	7	7	32.20	42.60

Figure 98: Changed Demands

Simulation Scenarios

Simulation scenarios provide a quick and easy way to test the network's resiliency in various scenarios. For each run a failure report is automatically generated.

1. From the main menu, select **Simulation > Simulation Scenarios**.

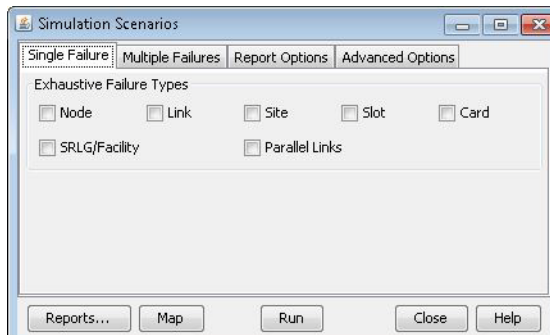


Figure 99: Failure Simulation Scenarios

Exhaustive Single Element Failure Scripts

1. To run an exhaustive single element failure for nodes and links, select the elements from the **Node** and **Link** tabs. This will fail one element, perform failure simulation, bring that element back up, and then fail the next element until all elements run through simulation.
2. The failure simulation will automatically create a peak utilization report indicating the worst utilizations for each link, across all single element failures, and the single element failure event that caused it. To report on the event which triggered the peak utilization for all periods of the **trafficload** file, go to the **Report Options** tab, and check "Include Simulation Events for all Periods in Peak Utilization Report".

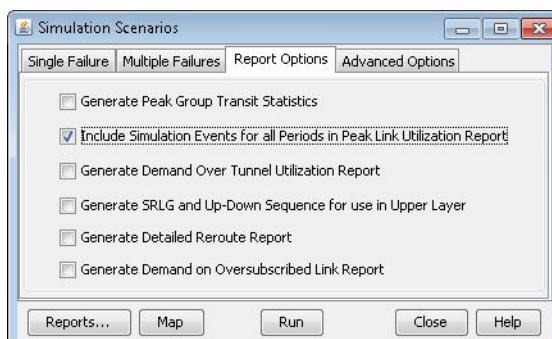


Figure 100: Report Options

3. Click the **Run** button after you selected the element type(s) to simulate. After the simulation is finished, a dialogue box provides a quick summary on the results. Click **OK** to continue.
4. From the main menu, select **Report > Report Manager > Simulation Reports** to view the simulation reports.
5. Report information will also be displayed on the console, one for each type selected. Each report will indicate the simulation sequence, simulation options, and simulation results.

Exhaustive Double and Triple Element Failures

1. To test scenarios where two or three elements fail simultaneously, select the **Multiple Failures** tab.
2. Select **Exhaustive Failure**, choose either Double or Triple for the number of simultaneous element failures, select the element type that you want to fail, and click **Run** to start simulation.

For example, selecting Double, Exhaustive, Node means: Fail two nodes simultaneously, perform failure simulation, bring those two nodes back up, select another set of two nodes, and repeat failure simulation until all node pair combinations have gone through simulation.

3. In the **Console**, every simulation event will be indicated followed by the tunnel and then demand statistics relating to maximum number of hops, average number of hops, number of demands failed, bandwidth of failed demands, and so on resulting from the event.
4. View the **Peak Utilization Report** in **Report Manager**.

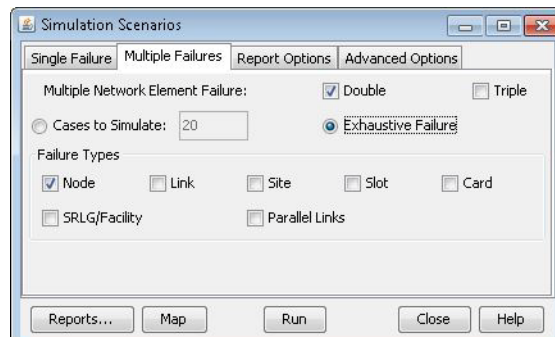


Figure 101: Multiple Failures Tab

Simulation Reports

To view simulation reports open the **Report Manager**.

Reports may be grayed out until running the appropriate **Simulation Scenario** and will be located under the Simulation Reports folder.

The generated reports and their respective filenames include the following:

- **Link Utilization Report (DVSIM)**: For each link, this report records the worst utilization that was encountered during the exhaustive failure simulation, and which particular failure event triggered it. See more details below.
- **Path Delay (PATHDELAY)**: For each demand, this report records the worst delay and worst distance that was encountered during the exhaustive failure simulation, and which particular failure event triggered it.
- **Simulation Overview (PeakSimSummary)**: Contains summary information about the failure simulations performed as part of the exhaustive failure simulation, including the simulation type (node, link, and so on.), element brought down, and the resulting statistics. These statistics include the number of demands/tunnels impacted, number of demands/tunnels failed (not including demands/tunnels terminating at failed nodes), maximum and average hop count after failure, number of links where bandwidth oversubscription occurred, and so on. For a detailed explanation of each field, view the header of the report.

- **Oversubscribed Links (PeakSimLink):** Link oversubscription details occurred during failure simulation are saved in PeakSimLink file. A utilization over 100% indicates that the traffic exceeds the link bandwidth.
- **Failed Path (PeakSimRoute):** The PeakSimRoute report indicates failed demand information, that is, demands which failed to be rerouted after they were connected, if any.
- **Rerouted Path (PeakSimReroute):** This requires enabling “Generate Detailed Reroute Report” from the **Simulation Scenarios Report Options** tab. The demand/tunnel reroute info is saved in PeakSimReroute file, indicating the demand that was disconnected, the original path, and the rerouted path.

Interpreting the Link Peak Utilization Report

The following is an example of a Link Peak Utilization Report from an exhaustive single node and link failure in the fish network. Some columns have been removed to highlight the key columns of this report.

Linkname	Anode	Znode	Type	TotalBw	UsedBw	PeakBw	PeakUtilPct	SimType	SimEvent	#Demand	PeakCnt	Cnt_SimType	Cnt_SimEvent
LINK7	CHI	DET	OC3	155.520M	101.079M	252.153M	162.14	LINEFAIL	LINK14	40	72	NDFAIL	WDC
LINK8	CHI	WDC	OC3	155.520M	130.676M	250.931M	161.35	LINEFAIL	LINK2	39	60	LINEFAIL	LINK2
LINK14	PHI	NYC	OC3	155.520M	149.686M	249.727M	160.58	LINEFAIL	LINK7	26	66	LINEFAIL	LINK7
LINK2	ATL	WDC	OC3	155.520M	173.584M	246.069M	158.22	LINEFAIL	LINK5	51	80	NDFAIL	CHI
LINK5	DAL	CHI	OC3	155.520M	105.868M	242.112M	155.68	LINEFAIL	LINK2	33	72	LINEFAIL	LINK2
LINK13	SJC	LAX	OC3	155.520M	172.688M	241.878M	155.53	LINEFAIL	LINK6	42	66	LINEFAIL	LINK6
LINK6	DEN	CHI	OC3	155.520M	67.912M	240.204M	154.45	LINEFAIL	LINK13	24	66	LINEFAIL	LINK13
LINK3	DET	BOS	OC3	155.520M	79.417M	228.834M	147.14	LINEFAIL	LINK14	34	60	NDFAIL	WDC
LINK9	HOU	DAL	OC3	155.520M	97.729M	223.267M	143.56	LINEFAIL	LINK2	43	76	LINEFAIL	LINK2
LINK7	DET	CHI	OC3	155.520M	90.070M	209.102M	134.45	LINEFAIL	LINK14	40	72	NDFAIL	WDC
LINK14	NYC	PHI	OC3	155.520M	117.849M	206.889M	133.03	LINEFAIL	LINK7	26	66	LINEFAIL	LINK7
LINK12	LAX	SDG	OC3	155.520M	86.112M	205.009M	131.82	LINEFAIL	LINK18	36	53	LINEFAIL	LINK18
LINK10	SFO	DEN	OC3	155.520M	36.402M	202.152M	129.98	LINEFAIL	LINK13	14	48	LINEFAIL	LINK13

Figure 102: Link Peak Utilization Report

For each link, this report records the **worst utilization** that was encountered, across all single node and link failures. For example, in the above report, the worst link utilization on LINK7 occurred during the failure of link Link14 (SimType LINEFAIL & SimEvent LINK14), with Link7 reaching a 162.14% Utilization (PeakUtilPct) as a result. Under normal circumstances, the utilization would be 101.079Mbps rather than 252.153Mbps.

This report also records the *peak number of demands that failed* on each link, across all single node and link failures. For example, the worst number of demands that failed on LINK7 was 72, and this event occurs from the failure of node WDC (Cnt_SimType NDFAIL & Cnt_SimEvent WDC). Under normal circumstances, the number of demands on this link would be 40.

If the network contains a trafficload file with the demands for each of 24 periods, there will be separate columns showing per period, the link's peak utilization across all single failures that were performed-- these columns are titled PeakLoadBW_1, PeakLoadBW_2, and so on., and can be compared to the regular utilization when there is no failure (LoadBW_1, LoadBW_2, and so on.). The worst values of the 24 periods will be saved in WrstPeakLoadBW and WrstLoadBW. The events triggering the peak utilization for each period will be indicated if the user checked the option to “Include Simulation Events for all Periods in Peak Utilization Report.”

Note that this report provides data for the logical links in the network. To obtain the physical port utilizations, select the **Peak Physical Link Utilization Report**.

As of 5.5.1, the format of this report has changed. Traffic information for every link is now displayed in two rows instead of one. The bandwidth and utilization information for one row are displayed for Anode, rather than both the Anode and the Znode. For backwards compatibility for scripting, add the following parameter to the dparam file, **changedvsimformat=0**.

Generating One Report Per Failure

Instead of showing just the worst case utilization per link for across all single failures, you may want to see a link's utilization for each failure. To generate a link utilization report per failure, use the text/ascii mode. This can be accessed either from an SSH/telnet window to the server, by executing the command `"/u/wandl/bin/bbdsgn spec_file_path"`.

1. For example, to generate a link utilization report per node failure, from the **BB DESIGN MAIN MENU**, select **8. Failure Simulation > 3. Failure Analysis > 3. Node Failure**
>Enter the runcode >Select Failure Report Options.

The following menu is displayed:

Report Generation Options:

1. Create one path placement report per simulation: No
2. Create one link utilization report per simulation: No

Select(q=Quit,?=Help):

2. Select 1 and 2 to create a report per simulation.
3. Enter 'q' to quit and start the simulations. After the simulations are done, quit out of all the menu by entering 'q' successively until returning back to the main menu. Enter 'q' again and 'y' to confirm.
4. Then check the resulting directory, and you should see one link utilization and path report per element failed.

Another example is to create a separate link utilization report for each individual facility failure. Run `/u/wandl/bin/bbdsgn <spec_file_path>`, substituting `<spec_file_path>` with the path of your specification file. Then select the following options: **8. Failure Simulation -> 3. Failure Analysis -> 5. Facility Failure**. Press Enter and then select a failure report option, **"2. Create one link utilization report per simulation"**. This would create one `LKUTIL.<facility_name>.<runcode>` report for each facility.

Viewing Simulation Results Graphically

After running a **Simulation Scenario** you can view the peak utilizations for each link from the Map using the **Utilization Legends > Peak Util** view.

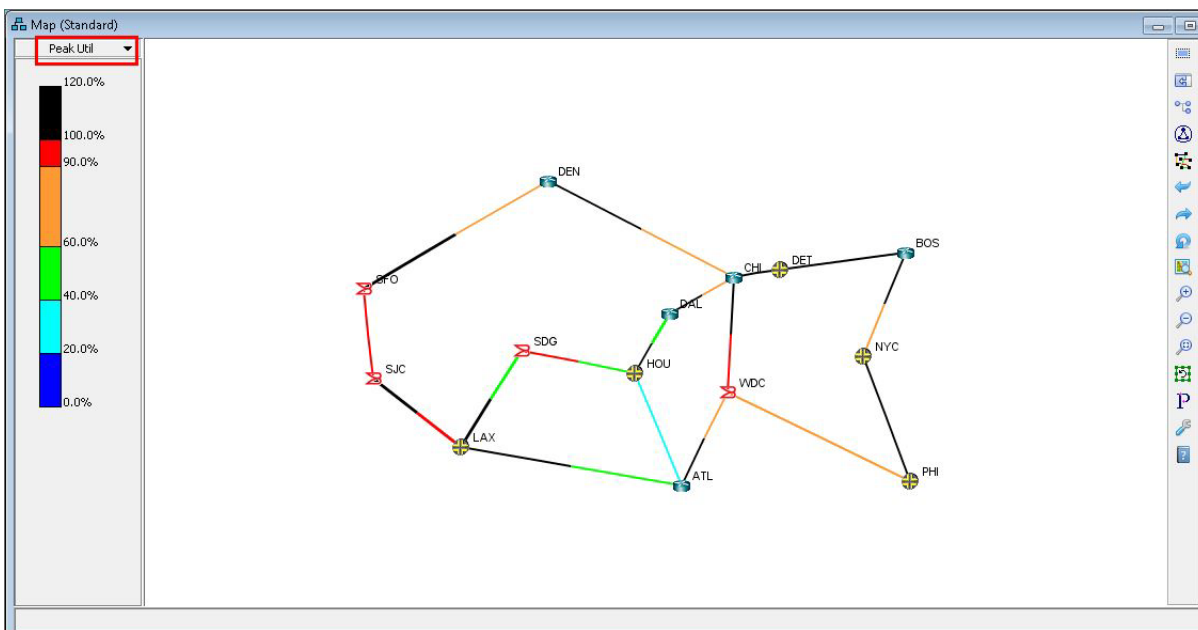


Figure 103: Peak Utilizations

Path Placement

This chapter describes path placement for demands. IP/MPLSView can be used to route unplaced demands, improve existing demand paths, or improve path diversity.

This module assumes that there is an existing backbone network, and tries to satisfy all demands and constraints through optimal calculation of the best assignment or layout. The constraints handled include: maximum number of link hops, media preference, diversity path constraints, and hardware bandwidth overhead calculations. This module can help the user evaluate whether the current network has enough bandwidth for new requirements entered in as demands. It can also be used to search for available paths in the backbone when the hardware fails to find a path.

If your hardware device supports the path select/path preferred or path required feature, you can have the hardware set up to use the path specifications generated by IP/MPLSView. The hardware then places the circuits according to your specification. Dynamic rerouting of circuits occurs only when the hardware fails. With the path select feature you have better control over the performance of the hardware. Customer experience has shown that a network is more reliable when the path select feature is used to route all the circuit paths according to the path design.

When to use

Use the path placement procedures to optimize your network demand paths, design diverse paths, or to perform what-if studies to see how demands are routed.

Prerequisites

Prior to beginning this task, you must have created a specification file as described in Chapter 5, Creating and Editing a Network Model or opened a specification file as described in Chapter 4, Opening and Closing a Network Project. You should then continue with these procedures to perform path placement.

Related Documentation

For an overview of IP/MPLSView or for a detailed description of each feature, refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Recommended Instructions

Following is a high-level sequential outline of the options available for path placement.

- Viewing Path Placement Options on page 124.
- Specifying Routing Methods on page 124.
- Defining Placement Order on page 125.
- Path Selection Method on page 125.
- Understanding Routing Paths on page 125.
- Network Grooming on page 125.
- Path Diversity Design on page 126.

Detailed Procedures

Viewing Path Placement Options

From the main menu, select the **Tools > Options > Design > Path Placement** option. Here you can specify various path placement parameters like the maximum number of hops for a demand path (Max Hop) and the oversubscription factor (for ATM).

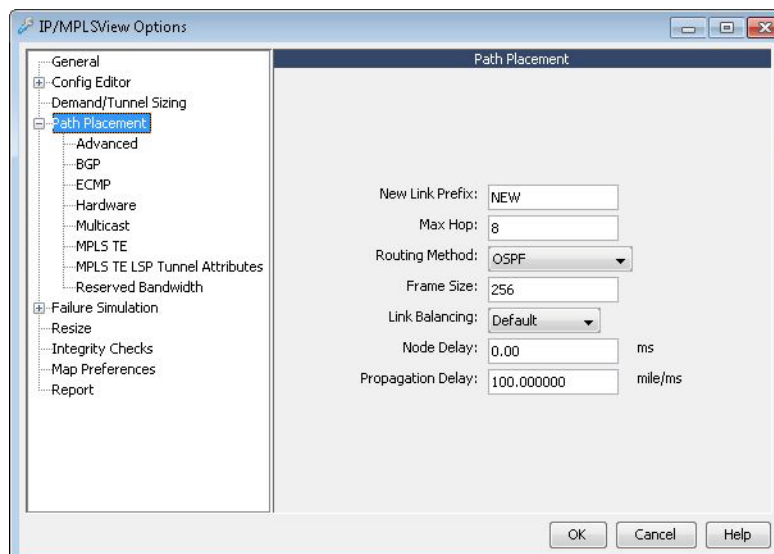


Figure 104: Path Placement Options

Specifying Routing Methods

Note that in cases in which routing methods differ per quality/class of service, the selection box may be on a different tab that allows you to specify a different routing method for each quality/class of service.

The options for routing method include Constant Distance (shortest hop), Actual Mileage, Admin. Weight, Delay, OSPF, ISIS, IGRP, and EIGRP. (Options may vary depending on the hardware type).

Defining Placement Order

To allow for a variety of possible hardware behaviors, eight different demand placement orders are currently supported: High-Priority First: Input Order, High-Priority First: Scramble, High-Priority, High BW Demands First: Input Order, Scramble Randomly, Low Bandwidth Demands First: Scramble, High Bandwidth Demands First: Scramble, Input Order. Here, input order refers to the order in the demand file.



Informational Note: If multiple demands are specified in one demand entry, they are routed one at a time, independently.

Path Selection Method

In the case where there are multiple shortest cost paths, three different path selection methods are currently supported: Hardware Implementation, Shortest Path Encountered First, and Select Randomly Among Best Candidates.

Understanding Routing Paths

Design from Scratch

When designing from scratch, all the existing paths are removed, and path placement is performed from scratch. From the main menu, select **Design > Demands > Route Paths > Route From Scratch**.

Tuning

When tuning demands, their paths are added if there is sufficient bandwidth and either the demand was previously unplaced or a better path has been found. From the main menu, select **Design > Demands > Route Paths > Tune All Paths** to place unplaced demands and improve upon existing placed paths. Or select **Design > Demands > Route Paths > Tune Unplaced Paths** to only place unplaced demands.

Restore Paths to Routing in Demand File

To reset the path placement to the contents of the demand file, from the main menu, select **Design > Demands > Route Paths > Restore**. This will allow you to perform a new tuning run starting from the original demand paths.

Network Grooming

Due to the dynamic rerouting of circuits during node or link failure, demand placement in the network may gradually become sub-optimal. The Network Grooming option looks at the current placement and the best placement of a demand (placement if it is the first demand placed). From the analysis report, you can see how optimal a demand path is. IP/MPLSView can then use this information to improve the path placement. There are several different link distance calculation methods that you can use to assess how optimal your demand paths are. Based on the results, you may wish to do one or more of the following: update the preferred path setting, change admin weights, and/or purchase additional links. See Chapter 18, Network Grooming for more details.

Path Diversity Design

This feature lets you design paths for diversity. Tunnels can be designed where its secondary or standby paths are node, site, link, or facility diverse from its primary path. Or two different tunnels can be designed to have diverse primary paths. Demands can use this path diversity design feature as well.

Route Optimization

This chapter describes how to study the impact of metric changes on demand routes.

When to use

Use these procedures if you want to study the impact of metric changes on the traffic routes.

Related Documentation

To automate link metric changes to reduce link congestion, refer to the metric optimization" chapter of the Router Feature Guide for IP/MPLS.

Recommended Instructions

Open a router network with OSPF or ISIS and run the Route Optimization wizard.

Detailed Procedures

The following steps indicate how to experiment with manual metric changes. To automate the process of metric optimization to reduce the worst link utilization, refer to the "Metric Optimization" chapter of the Router Feature Guide for IP/MPLS.

Using the Route Optimization Wizard

1. From the main menu, select **Design > Demands > Route Optimization**.
2. Click **Next** to continue.
3. The metric displayed is based on the protocol set in the **Options > Path Placement > Routing Method** field.
4. Select the links you would like to change and click **Modify Metrics**.
5. Enter the new metric values.
6. Click **Next** to make the changes.
7. The **Changed Routes** screen shows the effects of the metric changes.
8. You can save this information to a report by clicking **Report** or display the demand's path on the map by clicking **Show Paths**.

9. Click **Next** when you are ready to continue. This will bring up the **Save Changes** screen, in which you can decide whether or not to save the metric changes or to roll them back.

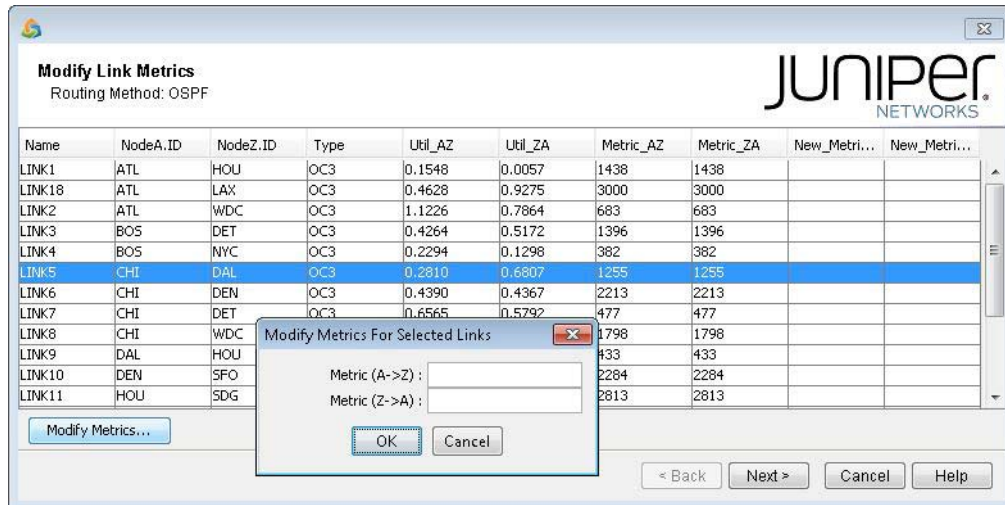


Figure 105: Modify Link Metrics

Network Grooming

This chapter describes the Network Groom feature in IP/MPLSView. The Net Groom tool allows the network operator to optimize the placement of a selected set of demands based on minimization of the total path cost of the associated demands.

When to Use

For many large networks and hardware configurations, when a demand is rerouted due to a network failure, the new route remains in use even when the network failure is resolved. Thus, after an extended period of time, a suboptimal set of routes may evolve in the network.

To re-establish an optimal set of routes for a network in this situation, IP/MPLSView has the Network Grooming feature.

Prerequisites

In order to use the Net Groom feature, you must first have a network design with nodes, links and demands placed. The demands may have been placed by a previous design or their paths altered due to network failures. Network Groom will find the optimal placement of demands using the current set of nodes and links in the network.



Informational Note: For MPLS-enabled router networks only: Network grooming of demand paths is not available if there are LSP tunnels in the network, as LSP tunnels can affect the routes of demands. However, in this case, network grooming can still be performed on *LSP tunnel paths* in Layer 2. The instructions in this chapter refers to demands and demand paths. However, the same applies to LSP tunnel paths.

Related Documentation

For more details about these windows, refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Recommended Instructions

1. Open the **Net Groom - Fetch Demands** window
2. Select the source nodes of demands whose paths will be considered for optimization.
3. Select the set of demands to be optimized based on Demand Type, Max BW, and Max Priority.
4. Set general options for Net Groom.

5. Click **OK** to open the **Net Groom** window.
6. Optimize demands.
7. Display the paths of the optimized demands in the **Map** window.
8. View a report of the optimized demands and paths.

Detailed Procedures

Once you have opened your network and verified that demands are placed, you may begin this section.

From the main menu, select **Design > TE Tunnels > Net Groom** (or **Design > Demands > Net Groom** for demands).

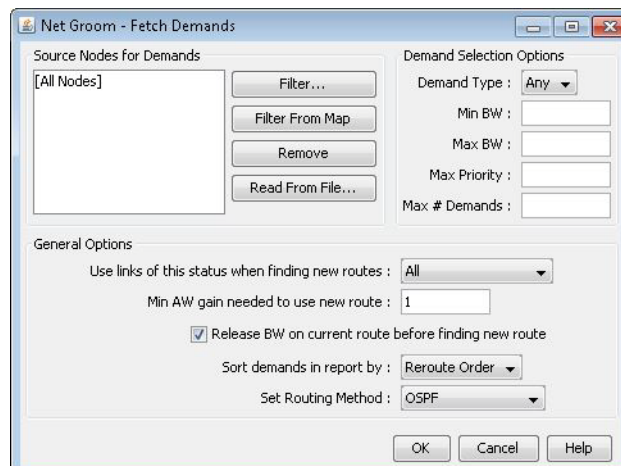


Figure 106: Net Groom - Fetch Demands

Selecting Demands for Net Groom

In the top left of the **Net Groom - Fetch Demands** window, you can select the source nodes of the demands you want to optimize. The default setting is all nodes.

Alternatively, source nodes for demands can be read in from a file.

Click the **Read From File** button and select the file from the server. The format of the file is simply the id of each node on a separate line. In this example, we want to optimize all demands, so we will use the default values.

In the top right of the **Net Groom - Fetch Demands** window, you can select which demands you want to optimize based on three selection criteria: **Demand Type**, **Max BW**, and **Max Priority**.

- If **Demand Type** is specified, only those demands that fit the demand type specified will become candidates for optimization.
- If **Max BW** is specified, only those demands with bandwidths that are less than Max BW will be chosen for optimization.
- If **Max Priority** is specified, only those demands that are of lower priority than Max Priority will be selected for optimization. If none of these fields are specified, then all demands are selected for optimization. In this example, we want to optimize all demands, so leave these three fields blank.

Setting Options for Net Groom

In the bottom of the **Net Groom - Fetch Demands** window, you can specify several options for Net Groom.

1. In the **Use links of this status when finding new routes** field, you may select **All**, **Live**, **Planned+Live**, or **New+Planned+Live**.
 - If **All** is selected, all links in the network will be used to route demands. If **Live** is selected, only links that are physically placed and active will be used to route demands.
 - If **Planned+Live** is selected, both active links and links that have been planned into the network will be used.
 - If **New+Planned+Live** is selected, then active, planned, and new (unplanned) links will be used to route the demands.

You can view and modify the status of a link by clicking on the **Location** tab in the **Links** or **Modify Link** window.

2. Here, we want to use all existing links to reroute the demands, so select **All**.
3. The **Min AW gain needed to use new route** field is used to specify how much change in admin weight is required in order for a demand to be rerouted when the optimization process is executed. If the admin weight gained from rerouting a demand is less than the value specified in this field, the demand will not be rerouted. If this field is left empty, all reroutes that result in an admin weight greater than zero will be implemented. We will leave this field set to 1.
4. The **Release BW on current route before finding new route** option sets whether or not the bandwidth used by the current placement of a demand will be ignored when the program is trying to find the optimal path for that demand. If this option is checked, the program will search for the optimal placement for the demand in the network without regard for the effect of the demand's current path on the network. If this option is not checked, the program will search for an optimal path for the demand in the network as if the demand's current path is still affecting the network.
5. To be conservative, we will uncheck this option.
6. The **Set Routing Method** allows you to specify which variables to use when optimizing demand placement.

- The **Sort demands in report by** field specifies how demands in the optimized links report will be sorted when a report is generated in the **Net Groom** window. If **Reroute Order** is selected, the demands will be sorted in the order in which they were rerouted. If **AW Gain** is selected, the demands will be sorted by AW Gain with largest AW Gain first.

Name	Node A	Node Z	BW	Orig AW	Best AW	Best AW Gain	Orig Path	Best Path	New Path
xflow18	HOUSTON	BOSTON	651.231K	7157	2880	4277	HOU--DAL--...	HOU--ATL--...	
flow25	BOSTON	WASHDC	46.458M	3671	759	2912	BOS--DET--...	BOS--NYC--...	
xflow24	SANJOSE	BOSTON	13.963M	8044	5132	2912	SJC--LAX--A...	SJC--LAX--A...	
xflow25	WASHDC	BOSTON	11.179M	3671	759	2912	WDC--CHI--...	WDC--PHI--...	
xflow23	SANFRANCI...	BOSTON	11.050M	8134	5222	2912	SFO--SJC--L...	SFO--SJC--L...	
xflow19	LOSANGELES	BOSTON	8.544M	7354	4442	2912	LAX--ATL--...	LAX--ATL--...	
xflow63	WASHDC	DETROIT	4.689M	5067	2155	2912	WDC--CHI--...	WDC--PHI--...	
flow24	BOSTON	SANJOSE	870.028K	8044	5132	2912	BOS--DET--...	BOS--NYC--...	
flow1	ATLANTA	BOSTON	730.017K	4354	1442	2912	ATL--WDC--...	ATL--WDC--...	
xflow22	SANDIEGO	BOSTON	662.011K	7617	4705	2912	SDG--LAX--...	SDG--LAX--...	
flow19	BOSTON	LOSANGELES	660.028K	7354	4442	2912	BOS--DET--...	BOS--NYC--...	
flow23	BOSTON	SANFRANCI...	660.028K	8134	5222	2912	BOS--DET--...	BOS--NYC--...	
xflow1	BOSTON	ATLANTA	660.028K	4354	1442	2912	BOS--DET--...	BOS--NYC--...	
flow63	DETROIT	WASHDC	651.231K	5067	2155	2912	DET--BOS--...	DET--BOS--...	
flow18	BOSTON	HOUSTON	558.028K	5792	2880	2912	BOS--DET--...	BOS--NYC--...	
flow22	BOSTON	SANDIEGO	558.028K	7617	4705	2912	BOS--DET--...	BOS--NYC--...	
xflow5	DETROIT	ATLANTA	441.231K	5750	2838	2912	DET--BOS--...	DET--BOS--...	

Total # of records : 28 records(start-end indices) : 1 - 28

Buttons: Optimize Selected Demands, Optimize All, View Paths, View Report..., Close, Help

Figure 107: Net Groom Window Before Optimization

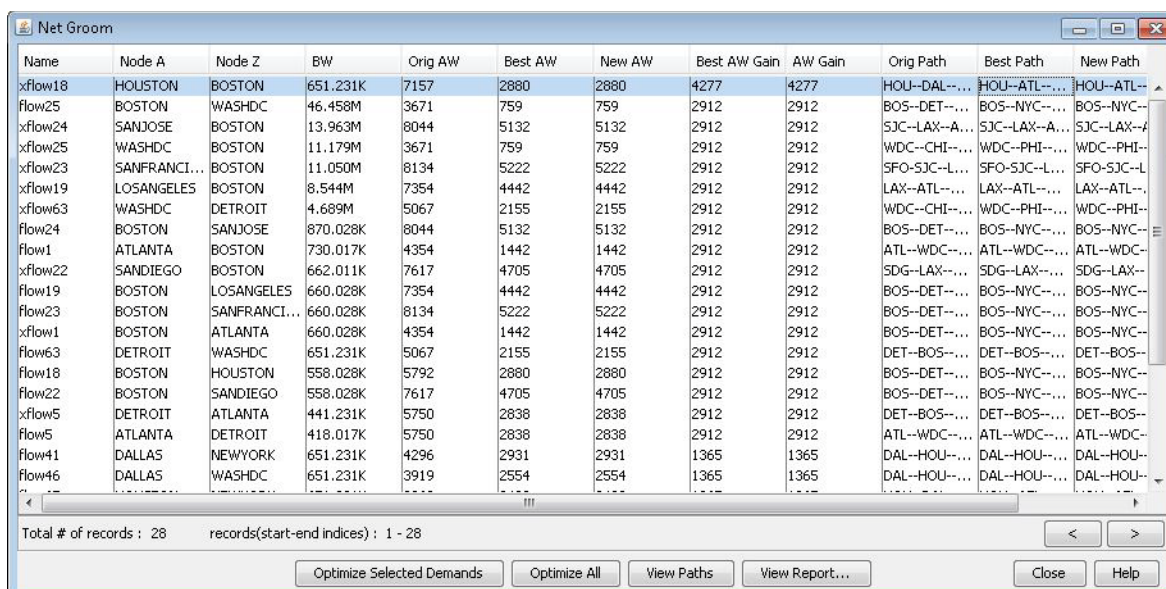
Optimizing Demands

The **Net Groom** window presents a table of available demands. Descriptions of each column are provided below.

Name	The Name of the demand
Node A	The source node of the demand
Node Z	The destination node of the demand
BW	Total bandwidth of the demand
Orig AW	The admin weight of the demand prior to optimization
Best AW	The best possible admin weight for the demand if there were no other demands in the network
Best AW Gain	The difference between Orig AW and Best AW
New AW	The optimal admin weight for the demand, taking into account the other demands in the network.
AW Gain	The difference between Orig AW and New AW
Orig Path	The original path of the demand
Best Path	The best possible path of the demand if there were no other demands in the network.
New Path	The optimal path for the demand, taking into account the other demands in the network.

- You may select multiple demands and click **Optimize Selected Demands** or you can **Optimize All** demands in the table.

- After you have optimized the demands, the **New AW**, **AW Gain**, and **New Path** columns will be populated. The **New AW** and **AW Gain** values will always be less than or equal to the **Best AW** and **Best AW Gain** values, since the latter two values are calculated for each demand assuming that there are no other demands in the network. When the existing demands are taken into account, the **Best AW** and **Best AW Gain** values cannot always be achieved. Thus, the best possible realistic admin weight and admin weight gains are displayed in the New AW and AW Gain columns.

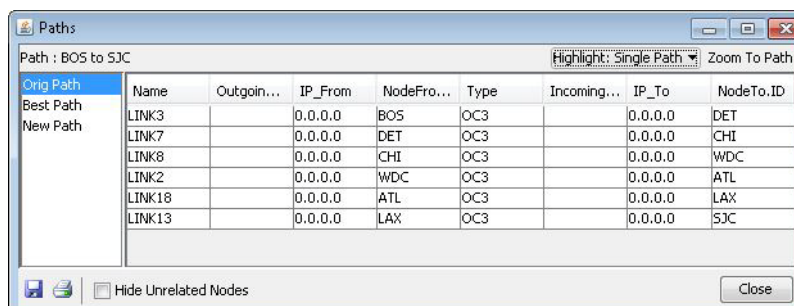


The Net Groom window displays a table with 12 columns: Name, Node A, Node Z, BW, Orig AW, Best AW, New AW, Best AW Gain, AW Gain, Orig Path, Best Path, and New Path. The table contains 28 records. The bottom of the window shows 'Total # of records : 28' and 'records(start-end indices) : 1 - 28'. Buttons at the bottom include 'Optimize Selected Demands', 'Optimize All', 'View Paths', 'View Report...', 'Close', and 'Help'.

Name	Node A	Node Z	BW	Orig AW	Best AW	New AW	Best AW Gain	AW Gain	Orig Path	Best Path	New Path
xflow18	HOUSTON	BOSTON	651.231K	7157	2880	2880	4277	4277	HOU--DAL--...	HOU--ATL--...	HOU--ATL--...
flow25	BOSTON	WASHDC	46.458M	3671	759	759	2912	2912	BOS--DET--...	BOS--NYC--...	BOS--NYC--...
xflow24	SANJOSE	BOSTON	13.963M	8044	5132	5132	2912	2912	SJC--LAX--A...	SJC--LAX--A...	SJC--LAX--A...
xflow25	WASHDC	BOSTON	11.179M	3671	759	759	2912	2912	WDC--CHI--...	WDC--PHI--...	WDC--PHI--...
xflow23	SANFRANCI...	BOSTON	11.050M	8134	5222	5222	2912	2912	SFO--SJC--L...	SFO--SJC--L...	SFO--SJC--L...
xflow19	LOSANGELES	BOSTON	8.544M	7354	4442	4442	2912	2912	LAX--ATL--...	LAX--ATL--...	LAX--ATL--...
xflow63	WASHDC	DETROIT	4.689M	5067	2155	2155	2912	2912	WDC--CHI--...	WDC--PHI--...	WDC--PHI--...
flow24	BOSTON	SANJOSE	870.028K	8044	5132	5132	2912	2912	BOS--DET--...	BOS--NYC--...	BOS--NYC--...
flow1	ATLANTA	BOSTON	730.017K	4354	1442	1442	2912	2912	ATL--WDC--...	ATL--WDC--...	ATL--WDC--...
xflow22	SANDIEGO	BOSTON	662.011K	7617	4705	4705	2912	2912	SDG--LAX--...	SDG--LAX--...	SDG--LAX--...
flow19	BOSTON	LOSANGELES	660.028K	7354	4442	4442	2912	2912	BOS--DET--...	BOS--NYC--...	BOS--NYC--...
flow23	BOSTON	SANFRANCI...	660.028K	8134	5222	5222	2912	2912	BOS--DET--...	BOS--NYC--...	BOS--NYC--...
xflow1	BOSTON	ATLANTA	660.028K	4354	1442	1442	2912	2912	BOS--DET--...	BOS--NYC--...	BOS--NYC--...
flow63	DETROIT	WASHDC	651.231K	5067	2155	2155	2912	2912	DET--BOS--...	DET--BOS--...	DET--BOS--...
flow18	BOSTON	HOUSTON	558.028K	5792	2880	2880	2912	2912	BOS--DET--...	BOS--NYC--...	BOS--NYC--...
flow22	BOSTON	SANDIEGO	558.028K	7617	4705	4705	2912	2912	BOS--DET--...	BOS--NYC--...	BOS--NYC--...
xflow5	DETROIT	ATLANTA	441.231K	5750	2838	2838	2912	2912	DET--BOS--...	DET--BOS--...	DET--BOS--...
flow5	ATLANTA	DETROIT	418.017K	5750	2838	2838	2912	2912	ATL--WDC--...	ATL--WDC--...	ATL--WDC--...
flow41	DALLAS	NEWYORK	651.231K	4296	2931	2931	1365	1365	DAL--HOU--...	DAL--HOU--...	DAL--HOU--...
flow46	DALLAS	WASHDC	651.231K	3919	2554	2554	1365	1365	DAL--HOU--...	DAL--HOU--...	DAL--HOU--...

Figure 108: Net Groom Window After Optimization

- At this point, the optimal placement of the demands have been calculated, and their paths are displayed in the **Net Groom** window. You can sort the rows based on each individual column by clicking on the desired column header.
- You can view the Orig, Best, and New paths in the **Map** window by clicking the **View Paths** button.



The View Paths window shows the path for 'Path : BOS to SJC'. It includes a table with columns: Name, Outgoing..., IP_From, NodeFrom..., Type, Incoming..., IP_To, and NodeTo.ID. The table lists six links (LINK3, LINK7, LINK8, LINK2, LINK18, LINK13) and their corresponding IP addresses and node IDs.

Name	Outgoing...	IP_From	NodeFrom...	Type	Incoming...	IP_To	NodeTo.ID
LINK3		0.0.0.0	BOS	OC3		0.0.0.0	DET
LINK7		0.0.0.0	DET	OC3		0.0.0.0	CHI
LINK8		0.0.0.0	CHI	OC3		0.0.0.0	WDC
LINK2		0.0.0.0	WDC	OC3		0.0.0.0	ATL
LINK18		0.0.0.0	ATL	OC3		0.0.0.0	LAX
LINK13		0.0.0.0	LAX	OC3		0.0.0.0	SJC

Figure 109: View Paths Window

- You can also view a report of the optimized demands and paths by clicking **View Report**.
- Click **Close** when done to exit from **Net Groom**.

Report Manager

This chapter describes how to use the Report Manager.

When to use

Use the Report Manager to view detailed information about your network. For example, you can view information about demand path and diversity, link utilization, equivalent cost multiple paths, link partitions, and BGP integrity checks.

Prerequisites

Prior to beginning this task, you must have created or opened a specification file. This tutorial will use spec.mpls-fish from the /u/wandl/sample/IP/fish directory. For help opening a specification file, see Chapter 4, Opening and Closing a Network Project. For help creating a specification file, see Chapter 5, Creating and Editing a Network Model.

Related Documentation

This guide explains how to use IP/MPLSView to view reports. For a guide on how to interpret the reports, please see the File Format Reference for IP/MPLSView.

For an overview of IP/MPLSView or for a detailed description for each window, refer to the IP/MPLSView Java-Based Graphical User Interface Reference.

Recommended Instructions

Following is a high-level sequential outline explaining how to use the report manager and recommended procedures.

1. Find out where to change report options as described in Report Options on page 136.
2. Open the Report Manager as described in Navigating the Report Manager on page 136 and view report information in the Explanation window.
3. Save a report to the server and/or client in CSV, XML, and HTML formats, as described in Saving Reports on page 138.
4. Customize a report by selecting, sorting, and resizing columns and filtering report rows as described in Customizing the Report View on page 137.

Detailed Procedures

Report Options

From the main menu, select **Tools > Options > Report** to change report options. Here you can specify the runcode, report style, node and link identification formats, and the units to use for bandwidth, distance, and currency.

The **Link Reference** will influence whether a path is displayed as a sequence of links or a sequence of nodes.

The default **Report Style** for the graphical user interface is “CSV” for Comma-Separated format, and is necessary to view reports in the **Report Manager** in table format.

Click **Cancel** to close this window without making changes.



Informational Note: The **Formatted Text** report style may not have all the information that the CSV report has because of limitations caused by this report style.

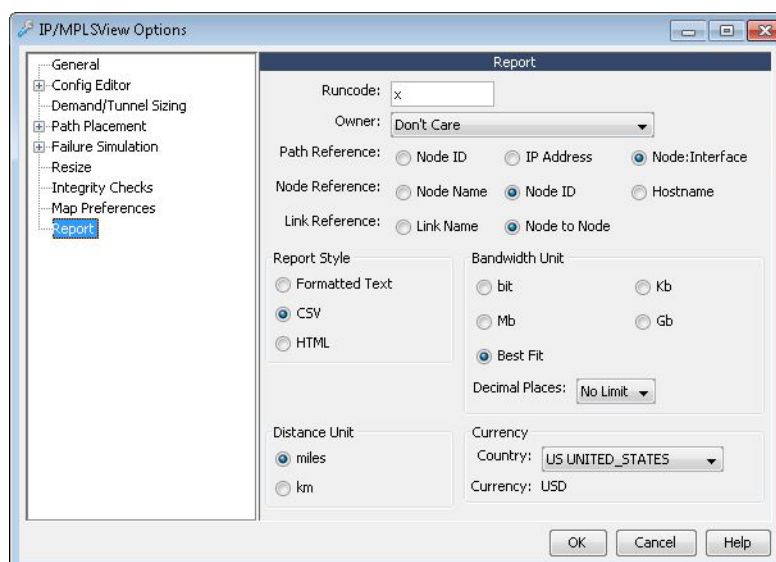


Figure 110: Report Options

Navigating the Report Manager

1. From the main menu, select **Report > Report Manager**.

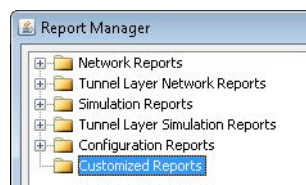


Figure 111: Report Manager

Note the contents may vary with the opened network's hardware type and license.

2. The main report categories include the following:
 - **Network Reports:** Details about demands, links, interfaces, group, nodes, network services, and network protocols.
 - **Tunnel Layer Network Reports:** Details about tunnel paths and diversity, demand traffic on tunnel, and tunnel RSVP BW.
 - **Simulation Reports:** Results of failure simulations. Some reports will not be activated until the appropriate failure simulation is run. For more information, see Chapter 15, Simulation.
 - **Tunnel Layer Simulation Reports:** Results of failure simulations related to the tunnel layer.
 - **Configuration Reports:** For router networks, the configuration reports include reports generated when importing in the router configurations and/or SNMP bridge files. For more information on the integrity checks report, see the *Router Feature Guide for IP/MPLS*, “Integrity Check Report” chapter. Additionally, SNMP, ISIS, OSPF, and additional status reports are generated.
 - **Customized Reports:** Contains user customized reports. Right-click the **Customized Reports** folder and select **Add a New Report** to open the customized report wizard.
3. Click the plus sign to expand the **Network Reports** folder to view network-related reports. Expand the **Demand Reports** folder to view the demand-related reports. Open the **Demand Path & Diversity Report** to see the actual path of demands in the network, and the resulting delay, number of hops, mileage and so on, for the demand.
4. Each page contains 200 lines. To see the next page, click the “Go to Next Page” button.
5. Click the plus sign to expand the **Link Reports** folder to view the link-related reports. Then click on the **Planned Link Utilization Report**. A file will automatically be created on the server side, and the contents will be displayed in a table in the right panel. This report gives you backbone link utilization based on demands loaded on the network. The console indicates that the file is saved in `LKUTIL.mpls-fish`.
6. Click the **Explanation** button above the report for information about the report, and descriptions for each column. Click **Close** to close this window.

By default, the reports are saved in the working network spec directory.

Customizing the Report View

1. Enter a query string in the “**Filter**” section of the **Report Manager**’s right pane to filter the rows of the Report Manager. This string can be a partial string.
2. Then click “**Search**” to limit the rows to those containing that string. Use “*” to retrieve all entries. (Note that this is case-sensitive.)
3. You can resize columns/rows and sort columns in the **Report Manager** as follows. To resize columns and/or rows, put your mouse pointer on the dividing line between two columns (or two rows) until the cursor becomes a double-arrow. Then press down the left mouse button key, and drag to the left or right (or top or bottom for row) to resize the column. To sort a column, click on the column header. If it is in descending order and you want ascending order, or vice versa, click on the column header again.

4. To limit the columns in view, right-click over the report on the left pane of the **Report Manager** and select **Select Columns** or click on the button **Select Columns** above the table. Columns in the **Unselected Column(s)** list can be moved to the **Selected Column(s)** list using the **Add->** button. Undesired columns can be removed by selecting them from the **Selected Column(s)** list and selecting the **<-Remove** button.
5. To reorder the columns, select one or more columns from the **Selected Column(s)** list and click the up or down arrows to change the position of the highlighted column(s).

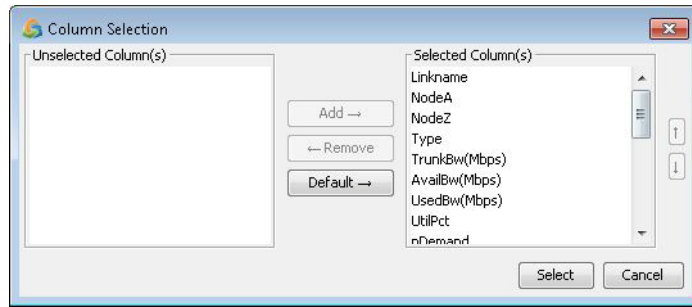


Figure 112: Select Columns Window

Saving Reports

- To save a report on your client machine, right-click over the report icon on the left pane and select **Save Whole Report to Client** to save the entire report to the client.
- To save only the rows that have been filtered by the Filter option below the report table, select **Save Filtered Report to Client**.
- Reports can be saved as CSV to your PC client by selecting **Export to Excel**.
- Reports can be saved as XML, CSV, or HTML by selecting **Convert Report** and choosing to save to either the server or PC client.
- For reports that have been saved on the server, the generated files may also be viewed from the **Report Viewer** if it is in CSV format. Go to the **File Manager** and navigate to the report. Right-click on that file and select **Open**.

Text Mode

This chapter briefly covers the use of the text mode in IP/MPLSView called **bdsn**, short for “backbone design”.

When to use

Use to run batch processes or operations where the graphical interface creates too much overhead in disk space or time. Text mode can also be used to automate certain processes.

Prerequisites

This topic requires a working understanding of Unix. You should also have a specification file containing the specification of your network files as explained in Chapter 3, Understanding the Specification File.

Recommended Instructions

Following is a high-level outline explaining how to use the text mode.

- Running Bdsn on page 140.
- Navigation in the menu on page 140.
- Read Files on page 141.
- Save Files on page 142.
- Saving Keystrokes for Batch Mode on page 144.

Detailed Procedures

Running Bbdsgn

1. To run the bbdsgn program, first go to the /u/wandl/bin directory using the following command:

```
$ cd /u/wandl/bin
```
2. Then enter the following at the command line prompt under the /u/wandl/bin directory to start bbdsgn:

```
$ bbdsgn <specfilepath>
```
3. where *<specfilepath>* is the relative or absolute path to the file that contains the names of the network input files. Or try:

```
$ cd <specfiledir>  
$ /u/wandl/bin/bbdsgn specfile
```

where *<specfiledir>* is the directory containing the specification file.
4. To view a help menu, enter:

```
$ /u/wandl/bin/bbdsgn -h
```

Navigation in the menu

When you execute “bbdsgn [specfile]”, where specfile is the file that contains the names of all the default input files, the following menu is displayed:

```
bbdsgn MAIN MENU (GENERIC):  
1. Read Files  
2. Save Files  
3. Network Information  
4. Reports  
5. Modify Configuration  
6. Path Placement  
7. Backbone Design  
8. Failure Simulation  
9. Discrete Event Simulation  
10. Call-by-Call Simulation  
11. SVC Call Simulation  
12. Access Design  
Select(q=QUIT, !=UNIX, g=Graphics):
```

Note that the options may vary depending upon the modules purchased.

To access a menu option, type in the corresponding number and then return. To get back to the main menu from a submenu, you can usually enter ‘q’ or simply **<Return>** to go up the menu levels, where the main menu is the top-most level. For example, type 5 and **<Return>** to select **5. Modify Configuration**.

To quit from the main menu, enter ‘q’ and **<Return>**. Then answer ‘y’ to the following question:

```
Exit BBDSGN? (y/n):
```



Informational Note: When the program is doing time-consuming work such as backbone design, failure analysis, or circuit path display, the user can interrupt the process by pressing <Ctrl>-c a few times. The user will be given the option of quitting the program at that point.

Read Files

Files specified in the specification file are read in automatically. Files not specified in the specification files may be read in using the **Read Files** option. This option can also be used to change these files for different design runs.

From the **Main Menu**, option 1 can be used to read in backbone demands, link configurations, the node weight file, and the user-defined link distance file.

1. To access this option, go to the main menu and select **1. Read Files** by typing '1' and pressing the Return key. The following menu is printed:

```
Read Files Menu:
  1. Demand    2. Link    3. Node Weight
  4. Node Parameter    5. User-Defined Link Distance
  6. PVC Load File=
Select (q=Quit, !=UNIX):
```

The options in this menu may also be different depending upon your hardware type.

2. When options 1, 2, 3, or 4 are selected, the bbdsgn program displays the current setting for that option and asks the user to enter a new filename, 'none', or to simply press <Return>. For example, select option 2 for the following:

```
Current bblink file = ./bblink.x
Enter Name of Link File or none: bblink.x2
```

3. The resulting options would be the following:
 - If you enter a filename, the contents of that file will be read. Note that when entering the file name, either the full path where the file is located must be entered or else the relative path from the current working directory.
 - If you enter 'none', the current information is removed. For example, if 'none' was entered at the backbone link file query, all the backbone links would be removed.
 - If you press the Return key, no action is taken. The file specified in the current settings has already been read in.
 - To reset the settings according to the contents of a file, the file name must be entered even if it is the same as the current file. The reason for this is that the contents of the file might have changed after it was initially loaded. For example, this could be the result of using the **'Modify Configuration'** option or manual edits to the file using a text editor.
4. If you do not remember the name of the file, press Return to get back to the prompt on the **Read Files Menu**, then type '!' and press Return to escape into a UNIX shell.

-
5. You may then use any of the following commands to list all the files in the working directory:

UNIX command	Corresponding action
ls	List contents of present working directory)
ls -l	List contents of present working directory in long format)
ls -lt	List contents of present working directory in long format and sort by time stamp)

6. To change to another directory, enter the command:

```
cd <directory>
```

7. Press CTRL-d or type “exit” to exit the UNIX shell.



Informational Note: The screen editor (for example, vi) should not be executed while in graphics mode.

Note that the options are slightly different for the demand file. Two demand files may be read in, a demand file and newdemand file. The newdemand file is used only if you have additional demands. This feature is intended to simplify the input process, since the program automatically merges the demands from both files if more than one file is supplied.

```
Current demand file (3418 demands)= ./demand.x
Current newdemand file= none
Replace Current Demand Requirements (default=no)? (y/n)      y

Demand File Name Modification Menu:
  1. demand file = ./demand.x
  2. newdemand file = none
Select (q=QUIT, !=UNIX): 1
-- Enter Name of demand file or none (default=./demand.x): demand.x2
Demand File Name Modification Menu:
  1. demand file = demand.x2
  2. newdemand file = none
Select (q=QUIT, !=UNIX):
```

Save Files

From the **Main Menu**, Option 2 is used to save the network files, among which include the demand path assignment, link configuration, user-defined link distance information, design parameter information. The complete design environment can also be saved to a directory using this option.

1. To access this option, go to the main menu and select **2. Save Files** by typing “2” at the main menu and then pressing Return. The bbdsgn program prompts you with the following menu:

```
Save Files Menu:
  1. Path and Link      2. User Link Distance
  3. Parameters         4. All
Select:
```

2. Demand information, link information, link distance information, and parameter information can be saved by selecting the corresponding menu item.

3. Enter the filename to which the information is to be saved, or simply press Return to not save it. Sample responses are given below. Notice that it is possible to selectively save the information (in this example, the link distance information is not saved).

```
Select: 1
Demands were read in from two demand files:
    679 demands from demand file(./demand.x)
    1709 demands from newdemand file(./newdemand.x)
Enter file name for demands: demand.new
>> Demand(s) saved to demand.new
Enter file name for newdemands: newdemand.new
>> Newdemands are saved to newdemand.new
Save Link Configuration to File: bblink.new
Replace DEF vendor by the vendor name selected(default=no)? (y/n)
>> Link Configuration saved to file bblink.new
```

The example also shows that when two demand files are read in, they are also saved in separate files.

(Note that after entering '1', the top three lines plus the lines involving the newdemand file will not be displayed if there is no newdemand file.)

4. To save the design parameters, select option 3. If a parameter file (dparam file) is indicated in the specification file, this file will then be treated as the default parameter file. To avoid overwriting the default parameter file, enter a new filename under which the information is to be saved. The following prompts are displayed when option 3 is selected:

```
Enter parameter file name (default=dparam.x): dparam.y
Parameter Settings saved in file dparam.y
View the file dparam.y(default=no)? (y/n):
```

5. Select option 4, 'All', for the following messages:

In order to save the design environment, a directory name and runcode must be specified. All related configuration and parameter files will be saved to the directory specified, with the runcode appended to the end of each file.

Enter q to quit

```
Enter directory name (default directory= /space/wandl/sample/fish): test
--- Directory test created
Enter runcode (default=x):
>> Design environment saved in directory /space/wandl/sample/fish/test
--- Runcode set to x
```

In the previous example, the statement 'default directory= /space/wandl/sample/fish' means the default directory is set to /space/wandl/sample/fish which is the current working directory.

To save the design environment in a directory other than the current one, specify the name of the destination directory. Note that if you want to use the same directory, you can press Return to accept the default in parentheses. This convention holds in general throughout the bbdsgn program.

Saving Keystrokes for Batch Mode

The keystrokes used to run `bbdsgn` can be traced to a file by the program. In the future you can then feed the keystrokes recorded in that file into `bbdsgn`.

In addition, you can create an input file manually and record the keystrokes you would like to use to run `bbdsgn`.

Tracing Input

To trace the input, navigate to the directory containing the specification file you want to run. Then type:

```
/u/wandl/bin/bbdsgn specfilepath traceinput
```

Substituting *specfilepath* with the relative or absolute path of your specification file. After your run, the file `input.trc` is created with your keystrokes recorded. To avoid overwriting this file in another trace input session, you can change the name of the file using the “mv” command. For instance,

```
mv input.trc trace1
```

Feeding Keystrokes

You can feed the keystrokes as an input using the “<” pipe. For example, if your specification file is `spec.uk` and your input trace file is named “`input.trc`”, you can type:

```
/u/wandl/bin/bbdsgn spec.uk < input.trc
```

To save the output, you can pipe the results into an output file using the “>” pipe. For example,

```
/u/wandl/bin/bbdsgn spec.uk < input.trc > myresults
```



Informational Note: If you are logging into the server remotely, and you wish for your process to run even if your connection is broken, use the “`nohup`” command.

```
nohup /u/wandl/bin/bbdsgn spec.uk < input.trc
```

Your results will be stored in “`nohup.out`”.

Creating Your Own Keystrokes Input

If you are familiar with the menu options and the flow of the menus, you can create your own input file.

For example, suppose in one scenario, we type in:

```
/u/wandl/bin/bbdsgn spec.uk traceinput
```

Then, suppose we choose the options necessary to perform a design, remove redundant links, perform a diversity design on `spec.dog`, and then exit from `bbdsgn`. Suppose the resulting `input.trc` file looks as follows:

```
7
3
Y
4
```



```
q
q
y
```

“7” was for the design menu, “3” was for basic design, “y” was for removing redundant links, and “4” was for diversity design. Finally, “q”, “q”, and “y” were used to exit bbdsgn.

Note that if instead of choosing “y” to remove redundant links, we simply pressed enter to choose the default value, the file would appear as follows:

```
7
3

4
q
q
y
```

Now, supposing we have a much larger set of demands to read in and we know the questions bbdsgn will ask will be exactly the same. We can create a second specification file spec.uk2 exactly the same except for a different demand file. Then we can run the following command:

```
/u/wandl/bin/bbdsgn spec.uk2 < input.trc > dog2output
```



Informational Note: Note that you must be careful in certain cases. If your input files are different enough that the bbdsgn program is triggered to ask an extra question or a different question somewhere, your input keystrokes may no longer be in response to the questions they were originally given in response to.

To see an example where the questions change and the input no longer gets used the same way, try the same command used for spec.dog for a specification file like /u/wandl/sample/fish/spec.mpls-fish. Because spec.mpls-fish is a router network, the questions are different. For example, one such question is,

```
Update demand routing tables (default=yes)? (y/n)
```

The corresponding input is the number 7, which was meant to select “Backbone Design” from the main menu. That no longer happens, since when the menu does appear, the corresponding input is the number 3 which takes you to the Network Information menu item.

