



Cisco Support Tools

User Guide

Release 1.0(1)

May 2004

Updated January 2006

Contains all information necessary to install, configure and use
Cisco Support Tools 1.0(1).

Table of Contents

About Cisco Support Tools Documentation	6
PART 1: SUPPORT TOOLS OVERVIEW.....	9
About Cisco Support Tools 1.0(1)	10
About Support Tools Components	12
About Support Tools Network Topology	14
About Support Tools Utilities	16
About the Support Tools Dashboard	24
About Support Tools Security	26
About Support Tools Platforms	27
Frequently Asked Questions About Support Tools.....	29
PART 2: INSTALLING AND CONFIGURING SUPPORT TOOLS.....	31
Section I. Installing Support Tools.....	32
About Installing Support Tools	33
How to Collect Information for a Support Tools Installation	37
How to Create Support Tools User Groups	39
How to Create the Distinguished User Account.....	41
About Support Tools Server Hardware and Software Prerequisites	43
How to Install the Support Tools Server.....	45
About Support Tools Node Hardware and Software Prerequisites	48
How to Install the Support Tools Node.....	49
How to Enable Ports for Support Tools.....	51
How to Test the Support Tools Installation	53
Section II. Configuring Support Tools	55
About Configuring Support Tools	56
How to Modify Support Tools Basic Configuration	57
How to Disable Continuous Virus Scan for the Repository	60
How to Configure the Dashboard Login for SSL.....	61
How to Modify the Process Information List	63
How to Configure SQL for Use with the System Interrogate Tool	66
Section III. Uninstalling/Reinstalling Support Tools	67
How to Uninstall Support Tools.....	68
How to Re-Install Support Tools	70
PART 3: USING THE SUPPORT TOOLS DASHBOARD.....	72
About Using the Dashboard for the First Time.....	73
How to Access the Support Tools Dashboard	75

How to Use the Select System Screen	76
How to Use the System Management Screen.....	78
How to End a Dashboard Session.....	80
About Dashboard Troubleshooting	81
PART 4: USING SUPPORT TOOLS UTILITIES	84
Section I. Using Cisco Web Utilities	85
About Cisco Web Utilities.....	86
Section II. Using Cisco Web Utilities from the Dashboard.....	87
How to Use the System Interrogate Screen.....	88
How to Use the Registry Screen	92
How to Use the Files Screen.....	94
How to Use the Compare Registries Screen	96
How to Use the Processes Screen	101
How to Use the Services Screen	103
How to Use the Log Collector	106
How to Use the Create Log Group Screen	109
How to Use the Log Groups Screen	111
How to Use the Create Log Collection Screen.....	114
How to Use the Log Collections Screen	116
Section III. Using Cisco Web Utilities from a Command Line	118
About Using Cisco Web Utilities from a Command Line.....	119
How to Use the Services Utility from a Command Line	123
How to Use the Processes Utility from a Command Line	127
How to Use the System Interrogate Utility from a Command Line.....	131
How to Use the Registry Utility from a Command Line	134
How to Use the Compare Registries Utility from a Command Line.....	138
How to Use the Log Groups Utility from a Command Line.....	143
How to Use the Log Collection Utility from a Command Line	147
Section IV. Using Cisco Common Utilities.....	152
About Cisco Common Utilities.....	153
How to Use the CICMan Utility	158
How to Use the CTITest Utility.....	160
How to Use the DBDiff Utility	165
How to Use the DumpCfg Utility	167
How to Use the ICMDBA Utility	169
How to Use the MPTrace Utility.....	170
How to Use the NICROI Utility.....	172
How to Use the NMStart Utility	177
How to Use the NMStop Utility	179

How to Use the OPCTest Utility.....	181
How to Use the Procmon Utility	186
How to Use the RTRTrace Utility	191
How to Use the RTTest Utility.....	192
How to Use the SS7NICTrace Utility	201
How to Use the VRUTrace Utility.....	202
Section V. Using Common Utilities	204
About Common Utilities.....	205
How to Use the Arp -a Utility.....	209
How to Use the CAT Utility.....	210
How to Use the CHMOD Utility.....	212
How to Use the CP Utility.....	214
How to Use the DF Utility	216
How to Use the Diff Utility	218
How to Use the DU Utility	221
How to Use the FGrep Utility	223
How to Use the Findstr Utility.....	225
How to Use the Grep Utility	228
How to Use the Head Utility	231
How to Use the IPConfig -all Utility	233
How to Use the ISQL/W Utility	235
How to Use the LS Utility.....	236
How to Use the MV Utility.....	239
How to Use the NBTStat Utility.....	241
How to Use the Net Session Utility.....	243
How to Use the Net Statistics Server Utility.....	244
How to Use the Net Statistics Workstation Utility	246
How to Use the NetStat Utility.....	248
How to Use the NSLookup Utility	250
How to Use the PathPing Utility	251
How to Use the Ping Utility	253
How to Use the PStat Utility.....	255
How to Use the RM Utility	257
How to Use the Route -print Utility	259
How to Use the Shutdown Tool Utility.....	261
How to Use the SQLEW Utility	262
How to Use the Stopshut Utility.....	263
How to Use the Strings Utility	264
How to Use the Tail Utility	266
How to Use the Touch Utility.....	268
How to Use the Tracert Utility	270
How to Use the WC Utility	271

How to Use the Which Utility.....	273
How to Use the WinMSD Utility.....	275
PART 5: REFERENCE.....	276
How to Stop and Start the Support Tools Dashboard/Tomcat.....	277
How to Stop and Start the Support Tools Server.....	278
How Stop and Start the Node Agent Service.....	279
How to Confirm the Support Tools Build Number.....	280
Cisco Support for Support Tools.....	281
Copyright.....	283
Index.....	285

About Cisco Support Tools Documentation

Welcome to the User Guide for Cisco Support Tools, version 1.0(1). This guide contains:

- An overview of Support Tools
- Installation and configuration instructions
- Instructions for using the Support Tools Dashboard
- Instructions for using Support Tools utilities from the Support Tools Dashboard
- Instructions for using Support Tools utilities from a command-line

Audience

This guide is intended for all Support Tools users, including individuals who deploy, install, administer, and use the product. It assumes that readers are familiar with:

- For Support Tools installation specialists: The platforms and operating systems on which Cisco ICM (Enterprise, Hosted, and IPCC) runs; basic Web server and firewall configuration.
- For Support Tools users: Cisco ICM and ICM components; the platforms and operating systems on which ICM runs.

The Support Tools Documentation Set

The following documentation is available for the Support Tools 1.0(1):

Document:	Primary Audience:	Description:
<i>Support Tools User Guide</i>	All Support Tools users	Provides information on: <ul style="list-style-type: none">• Support Tools features and deployment;• How to install and configure Support Tools;• How to use the Support Tools Dashboard;• How to use Support Tools utilities from the Support Tools Dashboard;• How to use Support Tools utilities from a command line; The Support Tools User Guide also provides context-sensitive Help for screens in the Support Tools Dashboard. Available formats: HTML and PDF
<i>Support Tools Release Notes</i>	All Support Tools users	The Support Tools Release Notes contain up-to-date information on known issues and workarounds and any special instructions not covered in this guide. Available formats: PDF

To Access Support Tools Documentation

After installation, to access the *Support Tools User Guide*:

- On the Support Tools CD, open `getstart.htm` at the top level of the CD.
- After installation, from the Support Tools Dashboard, click the Help icon.
- After installation, on the Support Tools Server, open `index.htm` (for HTML) or `icm_10_st_userguide.pdf` (for PDF) in `<icm_root_directory>\support_tools\documentation`.
- On a Support Tools Node, open `index.htm` (for HTML) or `icm_10_st_userguide.pdf` (for PDF) from `<icm_root_directory>\support_tools\documentation`.

Note: The Support Tools Release Notes are not installed with the product. They are only available on the Support Tools CD.

Support Tools documentation is also available for download from the technical publication section of Cisco's Web site at: www.cisco.com.

Support Tools Document Conventions

Support Tools documentation uses the following conventions:

Note	Indicates information of particular interest or significance.
Caution	Indicates the possibility of an adverse condition, such as poor or improper performance, data loss, or a security risk.
Courier font	Indicates a file or path name, user entry, or a code snippet.
< >	Indicates a variable. For example: <servername> represents a server's host name. When prefaced by "Press," a bracketed term represents a keystroke. For example, "Press <Enter>" means to press the Enter

Part 1: Support Tools Overview

About Cisco Support Tools 1.0(1)

Cisco Support Tools 1.0(1) is a suite of over fifty utilities that allow you to manage and troubleshoot the ICM servers which process call load, routing, and reporting. Through Support Tools, you can troubleshoot configuration and performance problems on these systems from a single machine in your network--the *Support Tools Server*.

Access to utilities in the Support Tools suite is through a browser-based interface--the Support Tools *Dashboard*--installed on the Support Tools Server. Levels of security control both access to the Dashboard and the ability to use specific tools once logged in. In low bandwidth conditions (for example, via dialup access) or when Web browsing is otherwise impractical, most Support Tools utilities can also be accessed and run via command line.

Key Features

The Support Tools suite includes the full set of standard diagnostic tools delivered with earlier ICM versions. It also provides key new functionality including:

- The ability to interrogate individual Support Tools nodes for their hardware/OS, Cisco component, and third party product information.
- The ability to view, stop, and start services running on Support Tools nodes.
- The ability to view and terminate processes running on Support Tools nodes.
- The ability to compare and synchronize registry settings from different Support Tools nodes.
- The ability to pull logs from most Support Tools nodes including ICM call routers, loggers, peripheral gateways (PGs), admin workstations (AWs), CTI Object Server (CTIOS), Cisco Collaboration Server (CCS), Cisco Email Manager (CEM), Cisco Media Blender (CMB), as well as Cisco CallManager.
- The ability to create enhanced time-synchronized merged logs across servers.

See Also

For related information, see:

[About Support Tools Components](#)

[About Support Tools Network Topology](#)

[About Support Tools Platforms](#)

[About Support Tools Security](#)

[About the Support Tools Dashboard](#)
[About the Support Tools Utilities](#)
[Frequently Asked Questions](#)

About Support Tools Components

Support Tools uses a client/server architecture to support remote ICM systems. The architecture is comprised of the *Support Tools Server* and the *Support Tools Node*. The server and node connect using TCP/IP and can exchange large volumes of messages with only a minimal impact to ICM system performance.

The Support Tools Server

The Support Tools Server is installed on a system that has little or no performance load, such as a standalone network server or a client ICM Admin Workstation (AW). To ensure redundancy, the Support Tools Server can be installed to multiple systems.

Major components of the Support Tools Server include:

- The Support Tools Dashboard, a browser-based interface that allows access to all Support Tools utilities.
- Cisco Web utilities-- a set of system management tools specifically designed for use with Support Tools.
- The Support Tools Repository, the central storage location for files (for example, logs, registries, etc.) gathered and saved through Support Tools. These files can be downloaded, viewed, and used for system comparison or restoration. Note that multiple installations of the Support Tools Server do not share a common repository--each will maintain its own instance.

The Support Tools Node

The Support Tools Node serves as the client to the Support Tools Server. It is installed to each ICM node to be managed through Support Tools.

Major components of the Support Tools Node include:

- The Node Agent Service, used to accept connections from the Support Tools Server.
- Command-line versions of Cisco Web utilities, allowing these tools to be run from a node when access to the Support Tools Server is unavailable or otherwise impractical (for example, during a network outage).
- A local Repository used to store saved and downloaded files when Support Tools utilities are run directly from a node (via a command line) outside of the Dashboard interface.

Note: The Support Tools Server installation automatically installs the Node Agent Service to the Support Tools Server, ensuring that this machine can also be managed through Support Tools.

See Also

For related information, see:

[About Support Tools 1.0\(1\)](#)

[About Support Tools Network Topology](#)

[About the Support Tools Dashboard](#)

[About the Support Tools Utilities](#)

[Frequently Asked Questions](#)

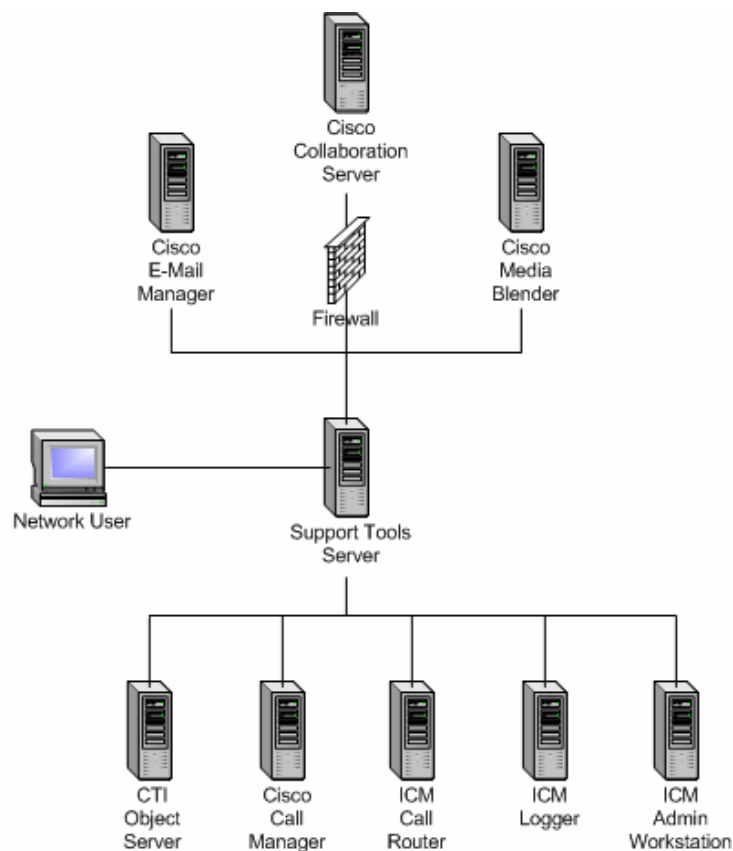
About Support Tools Network Topology

The Support Tools Server is installed to a standalone server or a client ICM Admin Workstation in your network. To ensure redundancy, the Support Tools Server can be installed to multiple systems. Note however, these multiple installations do not have the ability to share data with one another.

The Support Tools Server can communicate with Support Tools nodes through a firewall provided the appropriate ports on the firewall has been enabled.

User access to Support Tools is limited to your network. Remote users wishing to use the Support Tools Dashboard must first connect to the network--remote access via the Internet is not supported.

The following diagram illustrates a Support Tools deployment in a simplified ICM network.



See Also

For related information, see:

About Support Tools Components
About Support Tools Platforms
About Support Tools Security
How to Enable Ports for Support Tools

About Support Tools Utilities

Support Tools includes over fifty utilities to monitor, administer, and configure ICM systems on which the Support Tools Node is installed. Utilities are grouped into these categories:

- Cisco Web utilities: Four new Cisco utilities created for distribution with Support Tools 1.0(1).
- Cisco Common utilities: Pre-existing Cisco utilities which in most cases will already exist on ICM nodes as part of a standard ICM installation.
- Common utilities: Bundled third-party Windows, DOS, and Unix utilities. Windows and DOS utilities will already exist on ICM nodes as part of a standard Windows installation. Unix utilities are installed as part of the Support Tools node installation.

Utility Installation Locations

Support Tools utilities exist and can be run from individual ICM nodes (note that the Support Tools Server is itself treated as a Support Tools node). When you run one of these utilities through the Support Tools Dashboard, you are actually running the utility remotely from the currently selected node.

Not every utility is installed on every node. For the most part, Cisco Common Tools are installed on core ICM components only: AWs, PGs, Call Routers, and Loggers. For example, CICMan is only present on NAM Call Routers. See the table in the next section for more information.

If you attempt to launch a utility that is not present on the current system, the Support Tools Dashboard returns a message stating that the command is not recognized.

Non-Dashboard Utilities

Certain ICM and third-party utilities are only available for use through their own GUI; they have no command line access. This also precludes the ability to launch and run these utilities from the Support Tools Dashboard. Note that while these utilities are listed on the Dashboard menu, they cannot in fact be run from there.

These utilities include:

- icmdba
- isql/w

- rtrtrace
- sqlw
- ss7nictrace
- winmsd

Privileged Utilities

Some utilities, when accessed through the Support Tools Dashboard, can only be used by members of the Support Tools privileged user group. If you are not a member of the privileged user group, these utilities will not appear on the Support Tools Dashboard menu. These utilities include:

- ctitest
- icmdba
- nicroi
- nmstart
- nmstop
- procmon
- registry compare and repair
- rttest
- shutdown
- upctest
- Processes and services

Note that these privilege levels only apply to utilities when accessed through the Dashboard; they do not preclude users from running any utility through an independent command line on a node.

Command-Line vs GUI Access

For users who prefer command-lines to GUIs, note that in addition to access via the Support Tools Dashboard, most Support Tools utilities can also be run from an independent command line. For more information, see the instructions for individual utilities.

Also note that certain utilities (see *Non-Dashboard Utilities* above) have no command line access.

Support Tools Utilities List

The following tables list and describe the utilities accessible through Support Tools. Utilities are grouped into these categories:

- Cisco Web utilities
- Cisco Common utilities
- Common utilities

Cisco Web Utilities

The following Support Tools utilities were created by Cisco specifically for use with Support Tools 1.0(1). They can be run from within the Dashboard interface or from an external command line on either the Support Tools Server or an individual node.

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	External GUI - No Dashboard or Command-Line Access:
Log Collector	Use to collect and view log files from ICM nodes. Allows you to create merged logs from two ICM nodes for comparative analysis.	All Support Tools nodes		
Registry Compare and Repair	Use to view, compare and repair ICM node registry entries.	All Support Tools nodes	X	
System Interrogate	Use to display system information (for example, OS, hardware, ICM, SQL).	All Support Tools nodes		
Processes and Services	Use to view and stop Processes and services running on an ICM node.	All Support Tools nodes	X	

Cisco Common Utilities

The following Support Tools utilities were created by Cisco specifically for use with ICM. Most can be run both from within the Dashboard interface, or from an external command line.

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	External GUI - No Dashboard or Command-Line Access:
cicman	Use on a NAM CallRouter to view and set NAM to CICM session information.	NAM Call Routers		
ctitest	Use for debugging or demonstration in a Cisco ICM CTI environment. Can be used in place of an Interactive CTI client application.	CTIOS PGs	X	
dbdiff	Third-party (Microsoft) tool to compare database tables. Invoked with the diffconfig.bat tool.	Nodes on which SQL Server is installed		
dumpcfg	Dump Config ICM audit utility. Use to analyze the ICM Config_Message_Log table. Allows you to determine WHO did WHAT, WHEN, from WHERE, using WHICH program.	Loggers (but can be run from any ICM component)		
icmdba	ICM Database Administration tool. Use to create, monitor, and edit ICM databases, including Logger, HDS, and AWDB databases. ICMDBA is also used to manage various SQL Server operating parameters.	AWs, Loggers (ICM components with DBs)		X
mptrace	Use to provide a playback from the Nortel Meridian Automatic Call Distributor (ACD) to troubleshoot potential issues.	MerPim PGs		
nicroi	NIC Remote Operator Interface. Use to configure and debug the (old-style DOS-based) AT&T NIC. Often invoked with the NIC.BAT batch program.	Routers for customers running an AT&T NIC	X	
nmstart	Node Manager Start. Use to start an ICM service on a CallRouter, Logger, AW, or PG.	AWs, PGs, Call Routers, Loggers	X	
nmstop	Node Manager Stop. Use to stop an ICM Service on a CallRouter, Logger, AW, or PG.	AWs, PGs, Call Routers, Loggers	X	

opctest	Use to interpret a Peripheral Gateways (PG) status, statistics, etc. It is also possible to enable specific debug tracing in the OPC process.	PGs	X	
procmon	General-purpose debugging tool. Can be used in conjunction with various Network Interface Controller (NIC) and Peripheral Interface Manager (PIM) processes. Use to verify status, set debug trace bits, etc.	PGs, Call Routers	X	
rtrtrace	Use to set various debug tracing on a CallRouter. The additional tracing is output to the .EMS log files, and viewed with the Log Collector.	Call Routers		X
rttest	Use to interpret a Call Router's events and states live without interruption to the running router processes. Rttest has several subroutines that allow viewing status, statistics, etc. It is also possible to enable specific debug tracing in the call router.	Call Routers (but can be run from any ICM component)	X	
ss7nictrace	Use to view and set various debug trace bits on a new style NT SS7 NIC PG.	PGs		X
vrutrace	Use to output tracing information from a Voice Response Unit (VRU) device and to and from its PIM process. VRUTrace allows capture and playback of session data.	VRU PGs		

Common Utilities

The following Support Tools utilities are bundled third-party Windows, DOS, and Unix utilities. Most can be run both from within the Dashboard interface, or from an external command line.

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	No Dashboard or Command-Line Access:
Arp-a	Use to find the media access control address of a host on the same physical network.	All Support Tools nodes		
cat	Use the <i>CAT</i> utility to display, print, and combine files. Equivalent of the DOS type command.	All Support Tools nodes		
chmod	Use to change file permissions. Equivalent of the DOS attrib command.	All Support Tools nodes		

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	No Dashboard or Command-Line Access:
cp	Equivalent of the DOS copy command.	All Support Tools nodes		
df	Use to view disk free information.	All Support Tools nodes		
diff	Use to compare two text files or two directories.	All Support Tools nodes		
du	Use to view disk usage information.	All Support Tools nodes		
fgrep	Use to search for text patterns within a set of files.	All Support Tools nodes		
findstr	Use to find strings in logs or text files. Similar to UNIX's grep command. Useful when parsing log files or other text files to find a snippet of text.	All Support Tools nodes		
grep	Use to search for text patterns within a set of files.	All Support Tools nodes		
head	Use to view user-defined number of lines from the start of a file.	All Support Tools nodes		
IPConfig -all	Use to get host computer configuration information, including the IP address, subnet mask, and default gateway.	All Support Tools nodes		
isql/w	Interactive SQL for Windows. Use to query and update Microsoft SQL Server databases.	Nodes on which SQL Server is installed		X
ls	Use to view directory listings. Equivalent of the DOS dir command.	All Support Tools nodes		
mv	Use to move files and directories. Equivalent of the DOS move or rename command.	All Support Tools nodes		
NBTStat	Use to troubleshoot NetBIOS name resolution problems.	All Support Tools nodes		
Net Session	Use to view information about all current client sessions with the current system.	All Support Tools nodes		
NetStat	Use to view network information for the current system, including protocol statistics and current TCP/IP network connections.	All Support Tools nodes		
Net Statistics Server	Use to view statistics for the local Server service.	All Support Tools nodes		

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	No Dashboard or Command-Line Access:
Net Statistics Works	Use to view statistics for the local Workstation service.	All Support Tools nodes		
NSLookup	Use to get the IP address and fully qualified DNS of the current system's DNS name server.	All Support Tools nodes		
PathPing	A route tracing tool that combines features of the ping and tracert commands with additional information.	All Support Tools nodes		
ping	Low-level connectivity test. Use to verify end-to-end TCP/IP network infrastructure.	All Support Tools nodes		
pstat	Use to list all NT or Win2K processes, their process ID (PID), memory and cpu utilization, etc.	All Support Tools nodes		
rm	Use to delete files and directories. Equivalent of the DOS delete, recursive del subdirectories commands.	All Support Tools nodes		
Route-print	Use to view the contents of the network routing tables.	All Support Tools nodes		
shutdown	Use to shut down a remote host. When used from the Dashboard, shutdown will automatically restart the host after 60 seconds.	All Support Tools nodes	X	
sqlw	SQL Enterprise Manager. Use to monitor and update SQL Server operating parameters, expand databases, etc.	Nodes on which SQL Server is installed		X
stopshut	Use to stop a system from shutting down due to an application or operating system failure.	All Support Tools nodes		
strings	Use to find printable strings in a binary file.	All Support Tools nodes		
tail	Use to view user-defined number of lines from end of a file.	All Support Tools nodes		
touch	Use to change the creation date/time for any file.	All Support Tools nodes		
tracert	Related to ping command. Use to show each intermediary Hop in the TCP/IP network. Useful in troubleshooting LAN and WAN faults.	All Support Tools nodes		
wc	Use to view a count of characters, words, or lines in a file.	All Support Tools nodes		

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	No Dashboard or Command-Line Access:
which	Use to view the name of the first file encountered while traversing a path or directory tree.	All Support Tools nodes		
winmsd	Windows NT Diagnostics. Use to gather detailed configuration information about computers running Windows NT.	All Support Tools nodes		X

See Also

For related information, see:

[About Support Tools 1.0\(1\)](#)

[About Support Tools Components](#)

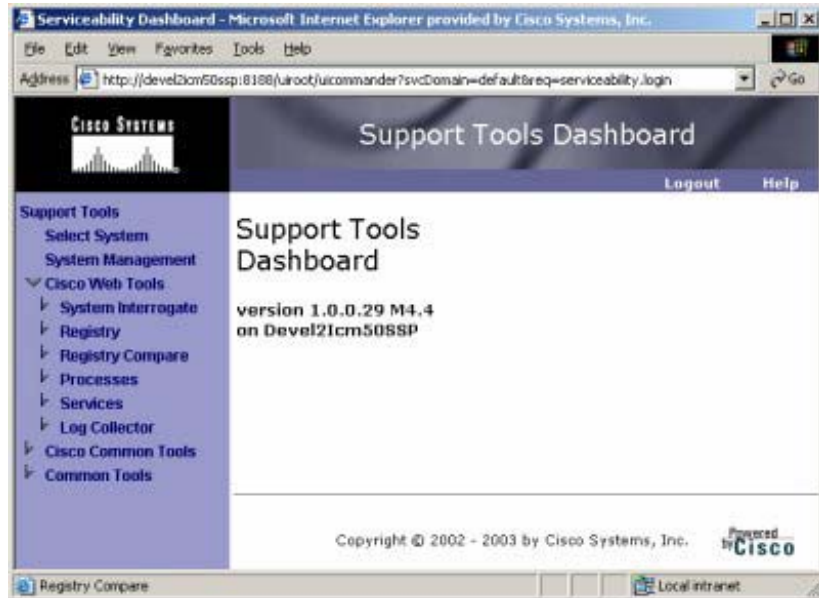
[About Cisco Web Utilities](#)

[About Cisco Common Utilities](#)

[About Common Utilities](#)

About the Support Tools Dashboard

The Support Tools Dashboard is a browser-based desktop that allows you to access and use all Support Tools utilities through a single interface. The Dashboard integrates access to all ICM systems, eliminating the need to manually connect to each remote system.



Accessing the Dashboard

Users access the Dashboard via a URL (`http://<hostname_or_ip_address>:<Tomcat_HTTP_port>`) entered in a browser window. Upon connecting, users must login by providing their Windows NT user name and password, or the user name and password of a privileged account.

Access to the Dashboard is restricted to the local network: access via the Internet is not supported. To access the Dashboard from a remote location, a user must connect and authenticate against the corporate network.

Dashboard Privileges

The Support Tools Dashboard recognizes two categories of users:

- *Privileged* users have full access to all Support Tools functionality, including tools and commands that can be used to modify system settings, registries, etc.
- *Regular* users have limited access to information gathering functionality only. Tools capable of system modification are not available to regular Support Tools users.

Support Tools assigns privilege based on regular Windows user groups (defined on your domain controller or through Active Directory). During the Support Tools Server installation, you are prompted to enter the name of one (and only one) Windows

user group that will serve as the Support Tools *privileged* group. Users in all other groups are treated as *regular* Support Tools users.

See Also

For related information, see:

[About Support Tools Components](#)

[How to Access the Dashboard](#)

[About Using the Dashboard for the First Time](#)

About Support Tools Security

Support Tools operates with the following security restrictions:

- A. Access to Support Tools is limited to the local network. Remote access via the Internet is not supported. Users must authenticate against the network in order to use Support Tools.
- B. Users must login to the Support Tools Dashboard using their network ID and password, or the ID and password of a valid network account. If desired, you can configure the Support Tools Login page to use SSL so that user passwords are encrypted when they are sent to the server.
- C. Within the Support Tools Dashboard, access to specific utilities is determined by Windows User Group membership. Only members of the user group designated as the Support Tools privileged group can use utilities capable of system modification from the Dashboard. Non-privileged users are limited to information gathering functionality only. Note that Support Tools utilities used outside of the Dashboard environment (namely, via command-line) do not impose these levels of privilege.

The use of Support Tools utilities outside of the Dashboard environment (accessed via a command line on a node) is not controlled by a login/password. System administrators can use Windows NTFS directory privileges to limit this method of access to specific users or groups.

See Also

For related information, see:

[About Support Tools Components](#)

[About the Support Tools Dashboard](#)

[How to Create Support Tools User Groups](#)

[How to Configure the Dashboard Login for SSL](#)

About Support Tools Platforms

This section provides platform support information for Support Tools 1.0(1), including:

- ICM Platform
- Support Tools Server Platform
- Support Tools Node Platforms
- Support Tools Dashboard Browser Support

ICM Platform

Support Tools 1.0(1) is designed for use with ICM version 5.0 and 6.0 (Enterprise, Hosted and IPCC).

Support Tools Server Platform

The Support Tools Server can be installed on a standalone server (that is, one on which no ICM products are installed), or on a client ICM Admin Workstation (AW) in your ICM network.

Caution: Due to possible performance issues, do not install the Support Tools Server on a distributor AW; AW installation should be limited to client AWs.

The Support Tools Server runs on the following platform:

Server Type:	Operating System:	Web Server/Servlet Engine:	Java:
Windows	Windows 2000 with service pack 3 or 4	Apache Tomcat 4.0.6 (included in Support Tools Server installer)	Java Runtime Environment 1.3.1 (included in Support Tools Server installer)

Support Tools Node Platform

The Support Tools Node can be installed on any ICM nodes listed below that you want to manage through Support Tools. All ICM 5.0 nodes run on the Windows 2000 Server sp3 platform:

- Cisco CallManager (CCM)
- Cisco Collaboration Server (CCS)
- Cisco E-Mail Manager (CEM)
- Cisco Media Blender (CMB)
- CTI Object Server (CTIOS)
- ICM Admin Workstations (AWs)
- ICM Call Routers
- ICM Peripheral Gateways (PGs)
- ICM Loggers

Support Tools Dashboard Browser Support

The Support Tools Dashboard is designed to work with Microsoft Internet Explorer 5.5 SP 2 or greater. Other browser platforms are not supported.

See Also

For related information, see:

[About Support Tools Components](#)
[About the Support Tools Dashboard](#)
[About Installing Support Tools](#)

Frequently Asked Questions About Support Tools

This section gives brief answers to some common questions about Support Tools. All of the subjects addressed here are described in greater detail elsewhere in this guide.

What is the benefit of Support Tools?

Support Tools provides a common interface (the *Support Tools Dashboard*) to approximately fifty utilities you can use to examine and troubleshoot Cisco ICM components.

What ICM version does Support Tools 1.0(1) work with?

Support Tools 1.0(1) is designed to work with ICM 5.0 and 6.0 (Enterprise, Hosted and IPCC). Earlier ICM versions are not supported.

What components make up Support Tools?

The *Support Tools Server* provides the engine and interface through which users access Support Tools utilities. The *Support Tools Node* is the listening agent that allows the Support Tools Server to interact with ICM components.

Where do I install the Support Tools Server?

The Support Tools Server can be installed on a client ICM Admin Workstation (AW) or (recommended) on its own dedicated network machine.

Caution: Because Support Tools Server installs the Apache Tomcat Web Server as a service, and with specific port requirements, DO NOT install it on a machine with a pre-existing installation of Tomcat in use by other applications.

Where do I install the Support Tools node?

The Support Tools node should be installed on each ICM component that you want to manage with Support Tools. See About Support Tools Platforms for a complete list of supported components.

What performance impact will running Support Tools have on my ICM system?

Support Tools processes run silently with minimum performance impact. It is recommended that: 1) the Support Tools Server be installed on its own dedicated machine, 2) because of the possible need to reboot, the Support Tools node be installed on ICM components during an off-production maintenance period, 3) large log collections be performed at times of low system traffic, and 4) the Support Tools Repository and its subdirectories be excluded from continuous virus scans.

Who can use Support Tools?

Any ICM user can use Support Tools. Permissions can be set to limit access to certain sensitive utilities in the Support Tools Dashboard to a privileged group.

What configuration does Support Tools require after installation?

While some optional configuration is available (see the Configuring Support Tools section) no post-installation configuration is required to use Support Tools out of the box.

Note, however, that to use Support Tools' merged log capabilities, each Support Tools Server and node must be able to communicate with the same Simple Network Time Protocol (SNTP or NTP) server. See How to Collect Information for a Support Tools Installation for more information.

See Also

For related information, see:

[About Support Tools Components](#)

[About Support Tools Platforms](#)

[About Support Tools Security](#)

[About Configuring Support Tools](#)

[How to Collect Information for a Support Tools Installation](#)

[How to Create Support Tools User Groups](#)

Part 2: Installing and Configuring Support Tools

Section I. Installing Support Tools

About Installing Support Tools

Installing Support Tools 1.0(1) consists of:

- A. Installing the Support Tools Server to a central server with access to your ICM system. If desired, you can install the Support Tools on multiple servers to provide redundancy. Note, however, that data is *not* shared between these separate installations.
- B. Installing the Support Tools Node on each ICM node to be managed through Support Tools. Note that the Node Agent is automatically included in the Support Tools Server installation; a separate installation of the node on this machine is not required.

Support Tools Installation Tasks

To install Support Tools, follow the sequence of steps in the checklist below. Each of these steps is described in greater detail in following sections. Cisco recommends you review this list before proceeding with the installation.

Note: Before proceeding with installation, also be sure to review the *Support Tools Release Notes* for any updates pertaining to Support Tools installation. The Release Notes are available at the top level of the Support Tools CD (`st_10_release_notes.htm`) and at <http://www.cisco.com>.

Caution: In some instances, the Support Tools Server and Node installations may require a reboot to complete. For this reason, it is strongly suggested that you perform the installation during a regular maintenance period when machines are not in production mode.

Step 1. Collect information for the install

Prior to installing Support Tools, collect the information that you will need to provide during the install.

Step 2. Create the Support Tools privileged user group

Create and assign users to the Windows user group that will serve as the Support Tools privileged user group.

❑ **Step 3. Create the Distinguished User Account**

The Support Tools Server installs the Tomcat Web server which runs as a service. This service must run as a distinguished user. To do this:

A. *Prior* to installing the Support Tools Server, create the Distinguished User.

B. *After* installing the Support Tools Server, assign the Distinguished User the requisite directory privileges.

❑ **Step 4. Review the Support Tools Server hardware and software prerequisites**

Prior to installing the Support Tools Server, ensure that the machine you are installing it on meets the minimum hardware and software prerequisites.

❑ **Step 5. Install the Support Tools Server**

The Support Tools Server installation includes:

- The Support Tools Dashboard
- The Support Tools Node
- Support Tools documentation
- Java Development Kit (JDK) 1.3.1
- Apache Tomcat 4.0.6

❑ **Step 6. Review the Support Tools Node hardware and software prerequisites**

Prior to installing the Support Tools Node, ensure that the ICM nodes you are installing it on meet the minimum hardware and software prerequisites.

❑ **Step 7. Install the Support Tools Node**

Install the Support Tools Node on each ICM node you want to manage through Support Tools. The Support Tools Nodes installation includes:

- The Support Tools Node Agent
- Support Tools Web utilities and certain Unix-style common utilities.
- Support Tools documentation

Note: Other Support Tools utilities are not part of the Support Tools installation. Rather, these utilities already exist on each node, distributed as part of standard ICM and Win2K installations.

Step 8. Enable ports for Support Tools

Support Tools uses the following ports:

- Tomcat (the Support Tools Web server) by default listens for HTTP requests on port 8188. It listens for HTTPS requests (used if you configure the Dashboard Login page for SSL) by default on port 8189.
- Support Tools requires a port (39100 by default) to communicate with ICM nodes. Ensure that this port is available on each node managed by Support Tools.

If a firewall stands between the Support Tools Server and any Support Tools node, and/or between the Support Tools Server and remote users, you must enable these ports on the firewall.

Step 9. Test the installation

After installing Support Tools, perform a few simple tests to verify that the install was successful.

Post-Installation Configuration

All basic configuration required by Support Tools is set during installation. Immediately after installation, no further configuration is necessary to begin using the product.

However, as desired, after installation you can modify the settings chosen during installation, or use certain optional behaviors (such as configuring the Dashboard Login page for SSL) that require further configuration. For more information, see [About Configuring Support Tools](#).

See Also

For related information, see:

[About Support Tools Platforms](#)

[How to Collect Information for a Support Tools Installation](#)

[How to Create Support Tools User Groups](#)

[About Support Tools Server Hardware and Software Prerequisites](#)

[How to Install the Support Tools Server](#)

[About Support Tools Client Hardware and Software Prerequisites](#)

[How to Install the Support Tools Node](#)

[How to Enable Ports for Support Tools](#)

[How to Create the Distinguished User Account](#)

[How to Test the Support Tools Installation](#)

[About Configuring Support Tools](#)

How to Collect Information for a Support Tools Installation

Installation Step 1

Prior to installing Support Tools, collect the following information that you will need to provide during the install:

For the Support Tools Server:

- A. Determine the name of the Support Tools privileged user group. Determine which users will be members of this group.
- B. Determine the username, domain, and password of the distinguished user for the Tomcat Web server.
- C. Determine the IP address or fully-qualified DNS of your network time server (see below for more).

For the Support Tools Node:

- A. Take an inventory of the ICM nodes on which you want to install Support Tools.
- B. Determine the IP address or fully-qualified DNS of your network time server.
- C. Determine whether a firewall stands between the Support Tools Server and any ICM nodes it will manage.

About the Network Time Server

To use Support Tools merged log capabilities, each Support Tools Server and node must be able to communicate with the same Simple Network Time Protocol (SNTP or NTP) server. This is required even for networks that have fully synchronized Windows 2000 systems as the allowable machine drift is greater than the tolerance of the log event times.

Note that Support Tools does not make time adjustment to host machines. Instead, the time offsets are tracked in a database to use for log merging only. Your SNTP or NTP server must support unauthenticated requests. Refer to RFC-2030 for more details on SNTP. Most Cisco IP Routers support the SNTP / NTP protocol. Consult with your IT department on available SNTP / NTP servers in your network.

If any system in a merged log collection cannot communicate with the SNTP server, the merge will fail, and the outputted log will contain the alert: "Missing SNTP data for all of the following files logs prevented merging logs: ..."

See Also

For related information, see:

[About Installing Support Tools](#)

[How to Create Support Tools User Groups](#)

[About Support Tools Server Hardware and Software Prerequisites](#)

[How to Enable Ports for Support Tools](#)

[How to Create Support Tools User Groups](#)

How to Create Support Tools User Groups

Installation Step 2

Support Tools recognizes two categories of users:

- *Privileged* users have full access to all Support Tools functionality, including tools and commands that can be used to modify system settings, registries, etc.
- *Regular* users have limited access to information gathering functionality only. Tools capable of system modification are not available to regular Support Tools users.

To Create Support Tools User Groups

Support Tools assigns privilege based on regular Windows user groups. This can be either a local group you define on the Support Tools Server machine or a domain group you define using Active Directory).

During the Support Tools Server installation, you are prompted to enter the name of one (and only one) Windows user group that will serve as the Support Tools *privileged* group. Users in all other groups are treated as *regular* Support Tools users.

Note: The privileged user group name is case-sensitive. It must match in case its corresponding Windows user group name.

Note: All Support Tools users must be members of the Windows Everyone group; anonymous users will not have access to Support Tools.

After installation, if necessary, you can modify the privileged group name by running the Support Tools Server configuration.

Creating Local Accounts on the Support Tools Server

To ensure you have local access to the Support Tools Dashboard in situations when network authentication is unavailable (for example, due to a downed network or authentication server) it is a good idea to create one or more local user accounts on the Support Tools Server.

To ensure full access to all utilities, these local accounts should be made members of a local user group named identically to your Support Tools privileged user group.

When logging into the Dashboard using a local account, enter `<hostname>\<Windows_username>` as your username; for example, `my_server\my_username`.

See Also

For related information, see:

[About Installing Support Tools](#)

[About Support Tools Utilities](#)

[How to Collect Information for a Support Tools Installation](#)

[About Support Tools Security](#)

How to Create the Distinguished User Account

Installation Step 8

The Support Tools Server installs the Tomcat Web server which runs as a service. This service must run as a distinguished user. To do this:

- A. *Prior* to installing the Support Tools Server, create the Distinguished User account as described below.
- B. *After* installing the Support Tools Server, assign the Distinguished User the requisite directory privileges as described below.

How to Create the Distinguished User Account

Before installing the Support Tools Server, on the machine on which the Support Tools Server will be installed, create a local or domain account to be used for the Tomcat HTTP service.

Note: This account is only used to run the Tomcat HTTP Service and should not be an administrator or user account for any other purpose. It is recommended that a local account be created since this will restrict access of the Tomcat HTTP Service to the one machine. In some scenarios using a local account for the Tomcat HTTP Service will prevent users from authenticating with a domain account. If this condition exists, you will need to use a domain account instead.

How to Assign Directory Privileges to the Distinguished User

If your Support Tools Server uses NTFS directory security that limits directory access to specific users, **after installing the Support Tools Server**, ensure that, in addition to the <machine>\System account, the distinguished user for the Tomcat HTTP Service has the following privileges to the Support Tools root directory and all of its subdirectories:

- Modify
- Read & Execute
- List Folder Contents
- Read
- Write

For example,

If the Distinguished User specified during installation is `my_domain\TomcatSvcAcct`, then `my_domain\TomcatSvcAcct` must be given the rights listed above to the Support Tools root directory and all of its subdirectories.

See your Windows documentation for instructions on granting NTFS security privileges.

About Support Tools Server Hardware and Software Prerequisites

Installation Step 3

The machine on which you install the Support Tools Server must meet the following requirements for:

- Hardware
- Software
- Network Access

Support Tools Server Hardware Prerequisites

The recommended minimum hardware configuration for your Support Tools Server is a Pentium-class PC with 512 Mb RAM and at least 50 Mb of available hard disk space.

You can install the Support Tools Server on a standalone server (that is, one on which no ICM products are installed), or on an existing ICM Admin Workstation (AW) in your ICM network.

Caution: Due to possible performance issues, do not install the Support Tools Server on a distributor AW; AW installation should be limited to client AWs.

Support Tools Server Software Prerequisites

Prior to installing the Support Tools Server, the following prerequisite software must be installed and properly configured:

Prerequisite:	Version:	Notes:
Microsoft Windows 2000	Server (with service pack 3 or 4)	

Additional Software Prerequisites

The following additional software prerequisites are installed automatically with the Support Tools Server:

- Java Runtime Environment (JRE), version 1.3.1.

- Apache Tomcat 4.0.6

Caution: Because Support Tools Server installs the Apache Tomcat Web Server as a service, and with specific port requirements, DO NOT install it on a machine with a pre-existing installation of Tomcat in use by other applications.

Note: Optionally, if you plan to configure the Support Tools dashboard Login page to use SSL (so that passwords are sent encrypted) an appropriate server certificate must be installed and SSL configured in Tomcat.

Network Access to ICM Nodes

Ensure that the machine on which you will be installing Support Tools Server has network access and administration rights to each ICM node on which the Support Tools Client will be installed.

See Also

For related information, see:

About Installing Support Tools

How to Install the Support Tools Server

About Support Tools Components

How to Configure the Dashboard Login for SSL

How to Install the Support Tools Server

Installation Step 4

You can install the Support Tools Server on any server with access to your ICM system. Before installing the Support Tools Server, ensure that the machine meets the prerequisite hardware and software requirements.

Caution: In some instances, the Support Tools Server installation may require a reboot to complete. For this reason, it is strongly suggested that you perform the installation during a regular maintenance period when the machine is in off-production mode.

The Support Tools 1.0(1) CD includes a Server Install Wizard that installs:

- The Support Tools Dashboard
- The Support Tools Node
- Support Tools documentation
- Java Development Kit (JDK) 1.3.1
- Apache Tomcat 4.0.6

Caution: Because Support Tools Server installs the Apache Tomcat Web Server as a service, and with specific port requirements, DO NOT install it on a machine with a pre-existing installation of Tomcat in use by other applications.

To Install the Support Tools Server

Note: The installation instructions below describe only those installer dialogs that require user input.

To install the Support Tools Server:

1. Log into the machine on which you are installing the Support Tools Server under an account with *local* Windows Administrator privileges for that machine.
2. Insert the Support Tools CD in the server's CD-ROM drive.
3. From the Windows Start Menu, select Run.
4. Browse to and double-click the Support Tools Server executable file located at:
`Server\setup.exe`.
5. If the installer detects a previous, earlier installation of the JDK (less than 1.3.1), you will be prompted to confirm an overwrite. Click Yes.

6. Specify values for each of the following:

Destination Path: Specify the location to install the Support Tools Server, or accept the default: `c:\Cisco\CCBU\supporttools`. You can install the Support Tools Server to any location on the local machine.

Note: If you plan on using the Cisco Security Agent (CSA) you can only change the drive letter and must leave the remaining default path information. This is because CSA security rules are written specifically based on the path names.

Repository Base Directory: Specify the path to the Support Tools Server Repository. This is the location where log files and saved registries and processes lists are stored. You can specify any location on the Support Tools Server's physical drive. The default is `<destination_path>\Repository`.

TCP/IP Port: The port the Support Tools Server uses to communicate with Support Tools Nodes. The default is 39100.

Administration Group Name: The name of the Windows User Group that has access to Support Tools functions capable of system modification (non-administration group users can use Support Tools for information gathering only). The default is Administrators.

Note: The Administration Group name is case-sensitive. It must match in case its corresponding Windows user group name.

Distinguished User Information: Specify the domain or local machine hostname, username, and password of the account that the Tomcat HTTP Service will run as. For more information, see How to Create the Distinguished User Account.

HTTP and HTTPS Ports: Specify the ports that the Tomcat Web server should use for HTTP and HTTPS requests. The Support Tools defaults are 8188 (HTTP) and 8189 (HTTPS).

Network Time Server: Specify the static IP address or fully-qualified DNS of your network time server. Also specify the frequency (in minutes) with which this machine should reconcile its clock against the network time server.

Note: As necessary, these values can also be modified after installation.

7. When the installation is complete, click Finish.

Depending on your local machine's configuration, you may be prompted by the installer to reboot the machine to complete the installation. The Support Tools Server and Tomcat will start automatically upon reboot.

8. If you are not prompted to reboot, the Support Tools Server and Tomcat will start automatically and is ready for use.

See Also

For related information, see:

[About Installing Support Tools](#)

[About Support Tools Server Hardware and Software Prerequisites](#)

[About Support Tools Components](#)

[How to Modify Support Tools Basic Configuration](#)

About Support Tools Node Hardware and Software Prerequisites

Installation Step 5

The machine on which you install the Support Tools Node must meet the following requirements for:

- Hardware
- Software

Support Tools Node Hardware Prerequisites

The hardware requirements for ICM nodes are predetermined by the ICM software they run.

The Support Tools Node requires a minimum of 20 Mb of available disk space.

Support Tools Node Software Prerequisites

Prior to installing the Support Tools Node, the following prerequisite software must be installed and properly configured:

Prerequisite:	Version:	Notes:
Microsoft Windows 2000	Server (with service pack 3 or 4)	
Microsoft Internet Explorer	5.5	Recommended but not required. Used to access the Support Tools Dashboard and Support Tools documentation.

See Also

For related information, see:

About Installing Support Tools

How to Install the Support Tools Server

About Support Tools Components

How to Install the Support Tools Node

Installation Step 6

You can install the Support Tools Node on any ICM node that you want to manage through Support Tools. Note that the Node Agent is automatically included in the Support Tools Server installation; a separate installation of the node on this machine is not required.

Note: Before installing the Support Tools Node, ensure that the machine meets the prerequisite hardware and software requirements.

Caution: In some instances, the Support Tools Node installation may require a reboot to complete. For this reason, it is strongly suggested that you perform the installation during a regular maintenance period when the machine is in off-production mode.

The Support Tools 1.0(1) CD includes a Node Install Wizard that installs the:

- Support Tools node agent
- Support Tools Web utilities and certain Unix-style common utilities.
- Support Tools documentation

To Install the Support Tools Node

Note: The installation instructions below describe only those installer dialogs that require user input.

To begin the Support Tools Node installation:

1. Log into the machine on which you are installing the Support Tools Server under an account with Windows Administrator privileges.
2. Insert the Support Tools CD in the server's CD-ROM drive.
3. From the Windows Start Menu, select Run.
4. Browse to and double-click the Support Tools Node executable file located at:
nodes\setup.exe.
5. Specify values for the following:

TCP/IP Port: The port the Support Tools Server uses to communicate with Support Tools Nodes. The default is 39100.

Destination Path: Specify the location to install the Support Tools Node, or accept the default: `c:\Cisco\CCBU\supporttools`. You can install the Support Tools Node to any location on the local machine.

Note: If you plan on using the Cisco Security Agent (CSA) you can only change the drive letter and must leave the remaining default path information. This is because CSA security rules are written specifically based on the path names.

Network Time Server: Specify the static IP address or fully-qualified DNS of your network time server. Also specify the frequency (in minutes) with which this machine should reconcile its clock against the network time server.

Note: As necessary, this value can also be modified after installation.

6. When the installation is complete, click Finish.
7. Depending on your local machine's configuration, you may be prompted by the installer to reboot the machine to complete the installation. The Support Tools Node will start automatically upon reboot.

If you are not prompted to reboot, the Support Tools Node will start automatically and is ready for use.

See Also

For related information, see:

[About Installing Support Tools](#)

[About Support Tools Node Hardware and Software Prerequisites](#)

[About Support Tools Components](#)

[How to Modify Support Tools Basic Configuration](#)

How to Enable Ports for Support Tools

Installation Step 7

Support Tools has the following port requirements:

Tomcat HTTP and HTTPS Port

Tomcat bundled with Support Tools by default listens for HTTP requests on port 8188 and HTTPS requests on port 8189.

If a firewall stands between the Support Tools Server and any Support Tools node, and/or between the Support Tools Server and remote users, you must enable these ports on the firewall.

As desired, during installation or subsequent reconfiguration you can modify Tomcat to use different HTTP and HTTPS ports. Note, however, that if you install the Support Tools Server on an ICM AW, ports 80 and 443 (the standard HTTP and HTTPS ports) are likely already being used by the ICM WebView reporting tool.

Note: Enabling the HTTPS port on a firewall is only necessary if you configure the Dashboard Login page for SSL. For more information, see [How to Configure the Dashboard Login for SSL](#).

Support Tools Listening Port

Support Tools requires a port to initiate connections with and listen for responses from ICM nodes. By default, the Support Tools Server uses port 39100. As desired, you can specify a different port during installation, or during subsequent modification to the Support Tool Server's configuration settings.

As part of the Support Tools installation/configuration:

- Ensure that the port used by the Support Tools Server is available on the ICM nodes it manages.
- If a firewall stands between the Support Tools Server and any of its ICM nodes, you must enable that port on the firewall to allow a one-way outbound connection from the Support Tools Server.

See Also

For related information, see:

About Installing Support Tools

How to Test the Support Tools Installation

Installation Step 9

After installing Support Tools, you should test your installation by:

- Accessing the Support Tools Dashboard.
- Testing the connection to Support Tools Nodes.

To Access the Support Tools Dashboard

To access the Support Tools Dashboard:

1. Open a Web browser (ST supports IE 5.5 or greater) and in the Address line, enter: `http<hostname_or_ip_address>:<Tomcat_HTTP_port>`.

Note that as shipped with Support Tools, Tomcat's default HTTP port is 8188. This default can be modified during installation.

2. Press <Enter>. The Support Tools Dashboard Login screen opens.
3. In the Login Name field, enter `<Windows_domain>\<Windows_username>`. For example, `my_domain\my_username`.
4. In the Password field, enter your Windows password (case-sensitive)
5. Click Login.

To Test the Connection to a Support Tools Node

To test the connection between the Support Tools Server and a node:

1. Login to the Support Tools Dashboard.
2. On the System Management screen, add the node(s) you want to test to the system list.
3. On the System Management screen, use the Test Connection utility to confirm the connection to individual nodes.

See Also

For related information, see:

How to Use the System Management Screen

Section II. Configuring Support Tools

About Configuring Support Tools

All basic configuration required by Support Tools is set during installation. Immediately after installation, no further configuration is necessary to begin using the product.

However, at some point you may want to modify the settings chosen during installation, or use certain optional behaviors that require further configuration.

These post-installation configuration options include:

- Modifying the basic Support Tools setting specified during installation (network time server, privileged use group, Repository path, and TCP/IP port).
- Excluding the Support Tools Repository from continuous virus scans.
- Configuring the Dashboard Login page to use SSL (so that passwords are encrypted).
- Installing Windows Management Instrumentation (WMI) on ICM nodes running SQL 2000 so that this information can be reported on by the Support Tools System Interrogate utility.
- Adding information about specific processes and services that display in the Processes & Services screen.

See Also

For related information, see:

[How to Modify Support Tools Basic Configuration](#)

[How to Configure the Dashboard Login for SSL](#)

[How to Modify the Process Information List](#)

[How to Configure SQL for Use with the System Interrogate Tool](#)

[How to Disable Continuous Virus Scan for the Repository](#)

How to Modify Support Tools Basic Configuration

The basic configuration settings necessary to run Support Tools (network time server, privileged use group, Repository path, and TCP/IP port) are specified during installation.

At any point after installation you can modify these setting, as necessary. You can:

- Modify the Support Tools Server configuration
- Modify the Support Tools Node configuration

To Modify Support Tools Server Configuration

To modify configuration settings for the Support Tools Server after it has been installed:

1. Rerun the Support Tools Server installer (`Server\setup.exe`) from your Support Tools CD.
2. During the reinstallation, the Install Wizard displays the configuration settings (TCP/IP Port, Administrator Group Name, etc.) from the current installation, rather than the installation defaults. As desired, modify the current values for these options:

Destination Path: Specify the location to install the Support Tools Server, or accept the default: `c:\Cisco\CCBU\supporttools`. You can install the Support Tools Server to any location on the local machine.

Repository Base Directory: Specify the path to the Support Tools Server Depository. This is the location where log files and saved registries and processes lists are stored. You can specify any location on the Support Tools Server's physical drive.

Administration Group Name: The name of the Windows User Group that has access to Support Tools functions capable of system modification (regular users can use Support Tools for information gathering only).

Note: The Administration Group name is case-sensitive. It must match in case its corresponding Windows user group name.

Distinguished User Information: Specify the domain or local machine hostname, username, and password of the account that the Tomcat HTTP Service will run as. For more information, see How to Create the Distinguished User Account.

TCP/IP Port: The port the Support Tools Server uses to communicate with Support Tools Nodes.

Note: When changing this value, remember to ensure that the port you choose is available on all Support Tool Nodes, and that it is enabled on firewalls standing between the Support Tools Server and its nodes.

HTTP and HTTPS Ports: Specify the ports that the Tomcat Web server should use for HTTP and HTTPS requests.

Network Time Server: Specify the static IP address or fully-qualified DNS of your network time server. Also specify the frequency (in minutes) with which this machine should reconcile its clock against the network time server.

3. When the reconfiguration is complete, click Finish.

Note: Support Tools includes several files that can be customized by users after an initial installation. Because rerunning basic configuration overwrites these customized files, Support Tools automatically creates backup copies (with the extension .bak). These backups can be restored re-introduce customization.

For more information, see [How to Re-Install Support Tools](#).

To Modify Support Tools Node Configuration

To modify configuration settings for the Support Tools Node after it has been installed:

1. Rerun the Support Tools Node installer (`nodes\setup.exe`) from your Support Tools CD.
2. During the reinstallation, the Install Wizard displays the configuration settings (TCP/IP Port, Administrator Group Name, etc.) from the current installation, rather than the installation defaults. As desired, modify the current values for these options:

Destination Path: Specify the location to install the Support Tools Node. You can install the Support Tools Node to any location on the local machine.

Network Time Server: Specify the static IP address or fully-qualified DNS of your network time server. Also specify the frequency (in minutes) with which this machine should reconcile its clock against the network time server.

3. When the reconfiguration is complete, click Finish.

Note: Support Tools includes several files that can be customized by users after an initial installation. Because rerunning basic configuration overwrites these customized files, Support Tools automatically creates backup copies (with the extension .bak). These backups can be restored re-introduce customization.

For more information, see [How to Re-Install Support Tools](#).

See Also

For related information, see:

[About Configuring Support Tools](#)

[About Support Tools Components](#)

[How to Collect Information for a Support Tools Installation](#)

[How to Create Support Tools User Groups](#)

[How to Enable Ports for Support Tools](#)

How to Disable Continuous Virus Scan for the Repository

Support Tools writes a large number of files (.log, .txt, and .zip) to the Repository in the course of normal use. The quantity of these files, if used in conjunction with continuous virus scanning, could result in a negative impact on performance.

If this performance degradation is not acceptable, Cisco recommends that you restrict continuous virus scanning in the Repository directory and its subdirectories to known file types only (which should omit .log and .txt files from scanning).

If performance degradation persists, Cisco recommends that:

- A. You exclude the Repository directory and its subdirectories from continuous virus scans run on the Support Tools Server and Node machines. Instead, limit scans on these directories to a nightly basis.
- B. After disabling virus scanning for the Repository directory and its subdirectories, use NTFS directory security to limit full access to these directories to administrators, NT Authority\System, NT Authority\Service and any other distinguished users that require direct access to these directories.

Caution: Steps A and B above should be used in concert with each other; that is, do not disable continuous virus scanning without also implementing NTFS directory security on the affected directories.

The default location of the Support Tools Repository on both the server and node is <support_tools_root>\Repository.

See Also

For related information, see:

About Configuring Support Tools

About the Support Tools Dashboard

How to Access the Support Tools Dashboard

About Support Tools Security

How to Configure the Dashboard Login for SSL

If desired, for extra security, you can configure the Support Tools Dashboard Login page to use SSL. This ensures that user passwords are encrypted when they are sent to the Support Tools Server.

Note the only the Login page will use SSL. Once logged into the application, all other page requests are sent over HTTP.

To Configure the Dashboard Login for SSL

To configure the Dashboard Login page to use SSL:

1. Ensure that an appropriate server certificate is installed on your Support Tools Server machine, and that Tomcat has been configured for SSL.

Instructions on how to configure Tomcat for SSL are available at:
<http://jakarta.apache.org/tomcat/tomcat-4.0-doc/ssl-howto.html>

2. In a text editor, open the `web.xml` file located at:
`<support_tools_root>\UIServer\uiroot\WEB-INF\.`

3. Locate the following closing tag: `</web-app>`.

4. Just before that tag, enter the following:

```
<security-constraint>
  <web-resource-collection>
    <web-resource-name>IServer</web-resource-name>
    <description>
      This pattern matches the url to the login page
    </description>
    <url-pattern>/default/serviceability/secure_login.jsp</url-
pattern>
  </web-resource-collection>
  <user-data-constraint>
    <description>
      All patterns matching the pattern above will use
confidential
      data constraints, namely, all data will be encrypted over
SSL.
    </description>
    <transport-guarantee>CONFIDENTIAL</transport-guarantee>
  </user-data-constraint>
</security-constraint>
```

5. Save the file.

6. Restart Tomcat.
7. If a firewall stands between your Support Tools Server and Support Tools nodes and/or remote users, remember to enable the appropriate SSL port on the firewall. For more information, see [How to Enable Ports for Support Tools](#).

Note: After configuring the Login page for SSL, users can continue to specify HTTP in the URL path to the Dashboard Login page. Entering HTTPS is not necessary.

Similarly, users can continue to specify the HTTP port number in the URL. Entering the SSL port number is not necessary. User will be automatically redirected to the correct port.

Therefore, the URL to the Support Tools login page using SSL is identical to that used for non-SSL:

```
http://<hostname>:8188
```

See Also

For related information, see:

[About Configuring Support Tools](#)

[About the Support Tools Dashboard](#)

[How to Access the Support Tools Dashboard](#)

[About Support Tools Security](#)

How to Modify the Process Information List

The process information list stores information about specific processes and services. The information from this file is then displayed to Support Tools users when they use the Processes and Services utility.

For each process or service defined therein, the file includes:

- The name of the process
- Whether the process is known, unknown, or critical
- Explanatory text about the process

Out-of-the-box, the process information list includes information on the following processes:

- All Cisco processes are identified as Known.
- Certain critical Microsoft OS-related processes are identified as Critical.
- Certain SQL and common Microsoft processes are identified as Known.

Modifying the Process Information List

As desired, you can modify the process information list to add new information about processes or services. For example, if you are running a virus checker, you may want to tag it as critical.

To modify the process information list:

1. Use a text editor to open `processinfo.xml` located on the Support Tools Server at `<support_tools_root>\repository\system files\`
2. Using proper well-formed XML syntax, add or modify entries as follows:

Process Type: Known, Unknown or Critical. **Note:** Use only these designations. Others will not parse correctly. Designating a process as Critical will prevent Dashboard users from stopping it (on the Services screen).

Name: The name of the process or service. Can include, but does not require, an .exe extension.

Manufacturer: Optionally, the manufacturer of the process or service.

Help: Any explanatory information you wish to display about the process or service.

Example: `<process type="known" name="notepad.exe" manufacturer="Microsoft"><help>This is Microsoft's Notepad text editor</help>`

When editing the file, consider the following:

- Cisco processes and services *not* identified in `processinfo.xml` are nonetheless automatically detected by the Processes and Services utility and labeled as Known.
- Non-Cisco processes and services *not* identified in `processinfo.xml` are labeled as Unknown.
- Be aware that Support Tools Dashboard users will be unable to stop services and processes tagged Critical.
- Consider that there is value in leaving less important processes and services tagged as Unknown. It allows Support Tools users to bulk select these items for termination when troubleshooting a node.
- Be careful to use correct XML syntax when editing the file. Incorrect syntax makes the file unreadable to the Support Tools parser, causing all but Cisco processes and services to display as Unknown.

Distributing a Modified Process Information List

The current `processinfo.xml` file is automatically propagated from the Support Tools Server to individual Support Tools nodes when these systems are selected for use from the Dashboard. In this way, an updated version of the file is available to users who launch the Processes and Services utility directly from a node via command line.

Note however that there may be instances where you may want or need to manually copy an updated `processinfo.xml` file from the Support Tools Server to another machine: for example, if you are running multiple installations of the Support Tools Server, or if you have nodes that are rarely selected for use from the Dashboard, but on which users may run the Processes and Services utility from a command line.

See Also

For related information, see:

[About Configuring Support Tools](#)

[How to Use the Processes Screen](#)

[How to Use the Services Screen](#)

How to Configure SQL for Use with the System Interrogate Tool

In order for the Support Tools System Interrogate utility to access and report on an ICM system's SQL database information, it must have access to Microsoft's Windows Management Instrumentation (WMI). WMI is not part of a standard Windows SQL install, and in most cases will therefore need to be installed separately.

To Install WMI

To install WMI:

1. Obtain the SQL 2000 Server CD, or download WMI at:
<http://msdn.microsoft.com/downloads/list/wmi.asp>.
2. On each ICM system that has SQL installed (and with which you want to use the System Interrogate utility--(for example, loggers, AWs) run setup from the \x86\OTHER\wmi directory.

Note: For additional information on installing and using WMI, see the MSDN library at: http://msdn.microsoft.com/library/default.asp?url=/library/en-us/wmisdk/wmi/sql_for_wmi.asp

See Also

For related information, see:

About Configuring Support Tools

How to Configure SQL for Use with the System Interrogate Tool

Section III. Uninstalling/Reinstalling Support Tools

How to Uninstall Support Tools

As desired, you can uninstall the Support Tools Server or Node by following the steps below. Note that it is *not* necessary to uninstall either as a precursor to reinstalling the product.

This topic describes how to:

- Uninstall the Support Tools Server
- Uninstall the Support Tools Node

To Uninstall the Support Tools Server

Uninstalling the Support Tools Server removes the Support Tools Server, its associated utilities, and registry entries associated with Support Tools. Note that uninstalling the Support Tools Server does not automatically remove Repository files, Tomcat 4.0.6 and the JDK 1.3.1.

To uninstall the Support Tools Server:

1. Stop Tomcat.
2. Use the Windows Services panel or a similar services tool to stop the Support Tools Appserver service, listed as Cisco CCBU Support Tools Appserver.
3. From the Windows Start menu, select Settings > Control Panel > Add/Remove Programs > Cisco Support Tools 1.0(1).
4. Click Remove.
5. If desired, after running the uninstaller, use Windows Explorer or a similar file management tool to delete the `Support_Tools` directory.

To Uninstall the Support Tools Node

Uninstalling the Support Tools Node removes the Node Agent only--it does not remove utilities or other items that are part of a standard ICM installation.

To uninstall the Support Tools Node from an ICM node:

1. Use the Windows Services panel or a similar services tool to stop the Support Tools node service, listed as Cisco CCBU Support Tools NodeAgent.
2. From the Windows Start menu, select Settings > Control Panel > Add/Remove

Programs > Cisco Support Tools 1.0(1).

3. If desired, after running the uninstaller, use Windows Explorer or a similar file management tool to delete the `Support_Tools` directory.

Note: After removing Support Tools from an ICM node, you should also remove that node's entry in the Support Tools System List (in the Support Tools Dashboard).

See Also

For related information, see:

[How to Re-install Support Tools](#)

[How to Stop and Start the Support Tools Dashboard/Tomcat](#)

How to Re-Install Support Tools

As desired, you can re-install the Support Tools Server or the Support Tools Node, using the same steps employed for an initial installation.

The following guidelines apply to reinstalling both the Support Tools Server and the Support Tools Node:

- It is not necessary to remove a current install prior to re-installation. Both the Support Tools Server and Node can be re-installed over themselves. Note that re-installation *will not* remove any files (for example, logs, registries, etc.) that have been saved to the Repository).
- It is not necessary to stop the Tomcat Web server or any Support Tools-related services in preparation for a re-install.
- If re-installing over a current installation, the Install Wizard displays the configuration settings (TCP/IP Port, Administrator Group Name, etc.) from the current installation, rather than the installation defaults.
- Following a re-install, you may want to restore backup copies of any customized files (see next section).

Restoring Customized .bak Files After Re-installation

Support Tools includes several files that can be customized by users after an initial installation.

Because re-installing the product overwrites these customized files, Support Tools automatically creates backup copies (with the extension .bak). These backups can be restored after re-installation to re-introduce customization.

Affected files include:

File:	Typically customized to:	Location:
processinfo.xml	Include additional information on system services and processes for display in the Dashboard's Processes and Services screens.	<support_tools_root>\repository\system files\ (On Support Tools Server and Nodes).
server.xml	Configure Tomcat for SSL.	<support_tools_root>\jakarta-tomcat-4.0.6\conf\ (On Support Tools Server only).

File:	Typically customized to:	Location:
systemlist.xml	Include list of user-added Support Tools systems recognized by the Dashboard.	<support_tools_root>\repository\system files\ (On Support Tools Server only).
web.xml	Configure the Dashboard Login page for SSL.	<support_tools_root>\UIServer\uiroot\WEB-INF\ (On Support Tools Server only).

See Also

For related information, see:

About Installing Support Tools

Part 3: Using the Support Tools Dashboard

About Using the Dashboard for the First Time

This topic describes some basic, important information on using the Support Tools Dashboard that all new users should be aware of.

Accessing Utilities in the Dashboard

All Support Tools utilities are accessed from the Support Tools menu. The menu groups utilities as:

- **Cisco Web Tools:** A set of four new Cisco management and troubleshooting utilities designed specifically for use from the Support Tools Dashboard.
- **Cisco Common Tools:** A set of previously released Cisco management and troubleshooting utilities.
- **Common Tools:** A set of common third-party utilities useful in troubleshooting ICM components.

Note that if you are not a member of the privileged user group, certain utilities will not be available to you and will not appear on the Support Tools menu. See About the Support Tools Utilities for a list of these privileged tools.

Selecting a System to Work With

The first action you typically take after logging in to the Dashboard is to select the system (ICM node) that you want to work with. Most Support Tools utilities cannot be used until you first select a system.

Note: The exception to this is the Log Collector; it is not necessary to select a system prior to log collection.

Select the current system using the Select System utility. After you have completed working with that node, you can use the Select System utility to choose a different system to work with.

Adding a System to the System List

In order to select a system to work with, you may first need to add it to the Support Tools system list (if it is not already defined in the list). You can do this using the Select System or System Management screens. In order to add an ICM node to the list:

- 1) The Support Tools node must have been installed on the node, and...
- 2) You must know the node's DNS entry or IP address.

It is important to note that the Support Tools Server has no automatic system discovery capability. That is, it does not automatically detect and list machines in your network running the Support Tools node. These must be added manually to the system list.

Navigating and Refreshing Pages in the Dashboard

When using the Dashboard, it's important to navigate and refresh pages using only the controls in the Dashboard interface.

Avoid navigating and refreshing using controls belonging to your browser, specifically the browser Forward, Back, and Refresh commands. Dashboard pages expire as soon as a request for a subsequent page is made; therefore, using the browser's Forward and Back commands may return you to an expired page. Because the Dashboard uses a frameset, its pages may not refresh properly using the browser Refresh command.

See Also

For related information, see:

[About the Support Tools Dashboard](#)

[About Support Tools Utilities](#)

[How to Use the System Management Screen](#)

How to Access the Support Tools Dashboard

To access the Support Tools Dashboard:

1. Open a Web browser (ST supports IE 5.5 or greater) and in the Address line, enter: `http<hostname_or_ip_address>:<Tomcat_HTTP_port>`.
 - Note that as shipped with Support Tools, Tomcat's default HTTP port is 8188. This default can be modified during installation.
2. Press <Enter>. The Support Tools Dashboard Login screen opens.
3. In the Login Name field, enter `<Windows_domain>\<Windows_username>`. For example, `my_domain\my_username`.
4. In the Password field, enter your Windows password (case-sensitive)
5. Click Login.

Note: Support Tools access is limited to individuals logged into your network. Remote access via the Internet is not supported.

See Also

For related information, see:

[About Support Tools Components](#)

[About the Support Tools Dashboard](#)

[How to Create Support Tools User Groups](#)

How to Use the Select System Screen

Use the *Select System screen* to:

- Select the Support Tools node you want to work with
- Add a node to the system list

You must select a system before you can interact with it through utilities in the Support Tools Dashboard.

To open the Select System Screen, from the Dashboard menu, select Select System.

To Select the Current System

Selecting a system specifies the Support Tools node you want to work with. You must select a system before you can interact with it through the Support Tools Dashboard. This is required even to use utilities that connect to multiple systems, such as registry compare.

To select the current system:

1. To select a system from the existing System list, click the Select System radio button, and then highlight the system you want, OR...
To select a system not on the System list, click the Enter System Name radio button, and then enter the DNS entry or IP address for that node. Optionally, to also add this node to the System list, check Add To List Of Systems.
2. Click Select System. The node is thereafter displayed as the current system at the top of each Dashboard Screen. Also indicated are the ICM product type(s) installed on the system. Can include:
 - ICM (AWs, PGs, Call Routers, and Loggers)
 - Call Manager (CCM)
 - Collaboration Server (CCS)
 - CTIOS
 - E-Mail Manager (CEM)
 - Media Blender (CMB)

Note: If the system you attempt to select is not available, an error message displays. Check that: the node's DNS or IP address was entered correctly; the

network connection to the node is active; the Support Tools Node Agent Service is installed and running on the node.

To Add a Support Tools Node to the System List

Adding a node to the system list makes it available for quick selection during future Dashboard sessions. Adding a node to the system list also specifies it as the current system.

To add a new node to the system list:

1. Click the Enter System Name radio button, and then enter the DNS entry or IP address for that node.
2. Check Add To List Of Systems.
3. Click Select System. The node is displayed as the current system at the top of the screen.

Note: If the system you attempt to select is not available, an error message displays. Check that: the node's DNS was entered correctly; the network connection to the node is active; the Support Tools Node Agent Service is installed and running on the node.

Note: To add multiple nodes more quickly, use the System Management Screen.

See Also

For related information, see:

[About Using the Dashboard for the First Time](#)

[How to Use the System Management Screen](#)

How to Use the System Management Screen

Use the *System Management screen* to:

- Add Support Tools nodes to the Support Tools system list
- Delete nodes from the system list
- Test the connection between the Support Tools Server and a node

To open the System Management screen, from the Dashboard menu, select System Management.

To Add a Node to the System List

Adding a node to the system list makes it available for quick selection during future Dashboard sessions.

To add a new node to the system list:

1. In the Add Systems box, add the DNS entry or IP address of one or more Support Tools nodes.
 - Add each node on a separate line. Do *not* use a delineator (for example, a comma) to separate entries.
 - Note that entries are not validated for correctness.
2. Click the Add System Button.

To Test the Connection to a Node

Testing the connection tests the connection between the Support Tools Server and that node

To test the connection to a node:

1. In the System Management box, highlight the Support Tools node you want to test.
2. Click the Test Connection button.

If the system you attempt to test is not available, an error message displays. Check that: the node's DNS or IP address was entered correctly in the system

list; the network connection to the node is active; the Support Tools Node Agent Service is installed and running on the node.

To Delete a Node from the System List

To delete a Support Tools node from the system list:

1. In the System Management box, highlight the Support Tools node you want to delete from the list. Use Shift-Click or Control-Click to select multiple list items.
2. Click the Delete System button.

See Also

For related information, see:

About Using the Dashboard for the First Time

How to End a Dashboard Session

To terminate a Dashboard session, click the Logout link in the Dashboard. Dashboard sessions also time out automatically after 60 minutes after you close your browser.

Note that:

- Sessions will not time out as long as your browser remains open.
- Terminating a session will not terminate any Dashboard processes that may be running (for example, log collection). These will continue to run to completion.
- The Dashboard does not place a restriction on multiple logons. It is not necessary to terminate a session in order to begin a new one.

About Dashboard Troubleshooting

This topic lists some basic problems you may encounter accessing and using the Support Tools Dashboard and some possible workarounds.

Login/Connection Problems

The following table describes possible errors encountered when attempting to access or login to the Support Tools Dashboard.

Symptom:	Possible Cause:	Possible Workaround:
Unable to access Support Tools Login page AND Browser returns a Page Not Found (404) error.	1. Incorrect URL entered OR 2. Incorrect port entered in URL OR 3. Tomcat Web server not started on Support Tools server OR 4. Support Tools Server or LAN connectivity down.	1. Re-type URL: http://<hostname>:8188. 2. By default, Tomcat bundled with Support Tools uses ports 8188 (HTTP). Confirm with your administrator that this default has not been modified. 3. Confirm with your administrator that the Tomcat Web server is started and running properly on the Support Tools server. 4. Confirm with your administrator that the Support Tools server machine is up and running properly, and that LAN connectivity exists. For more information, see: How to Access the Support Tools Dashboard

Symptom:	Possible Cause:	Possible Workaround:
Can Access Dashboard Login Page but cannot log in	Incorrect or invalid username or password entered.	<p>Re-enter username and password as follows:</p> <p>Network Users:</p> <p>Name: <windows_network_domain>\<username></p> <p>Password: <network password></p> <p>Local Users:</p> <p>Name: <localhost>\<username></p> <p>Password: <local_password></p> <p>Note: Local users are typically administrators for whom Windows user accounts have been set up on the Support Tools Server machine. This allows them to access the Dashboard in scenarios when normal network authentication is unavailable.</p> <p>For more information, see: About Support Tools Security</p>

Utility Problems

The following table describes possible errors encountered when attempting to use Dashboard utilities following a successful login.

Symptom:	Possible Cause:	Possible Workaround:
Error returned when attempting to select a specific host as the current system	<p>1. The machine, or LAN connection to that machine, is down</p> <p>OR</p> <p>2. Machine is identified incorrectly in Support Tools System List</p> <p>OR</p> <p>3. Support Tools Node not installed on that machine</p> <p>OR</p> <p>4. Support Tools Node Agent Service stopped on that machine</p>	<p>1. Confirm that the machine is running and that LAN connectivity is available.</p> <p>2. Confirm that the computer name for that machine as defined in the Support Tools System List (displayed on the Dashboard's System Management screen) is correct.</p> <p>3. Confirm with your administrator that the Support Tools Node is installed on that machine.</p> <p>4. Confirm with your administrator that the Support Tools Node Agent Service is started on that machine.</p> <p>For more information, see: How to Use the System Management Screen</p>

Symptom:	Possible Cause:	Possible Workaround:
<p>Certain utilities are missing from the Dashboard menus (e.g., Services screen, Processes screen)</p>	<p>You are not a member of the Support Tools privileged user group.</p>	<p>Certain utilities are only available for members of the Support Tools privileged user group. Contact your administrator if you believe you should have access to these utilities.</p> <p>For more information, see: About the Support Tools Dashboard</p>
<p>Error Processing Request message returned when attempting to run a utility</p>	<p>1. That utility is not installed on the current system OR 2. Appserver service not stated on the Support Tools Server</p>	<p>1. Some utilities are only available for use on specific ICM components. Consult the online Help for that utility to determine if its use is limited to specific components.</p> <p>For more information, see: About Support Tools Utilities</p> <p>2. Confirm with your administrator that the Support Tools Appserver service is started on the Support Tools Server machine.</p> <p>For more information, see: How to Stop and Start the Support Tools Server</p>

See Also

For related information, see:

- [How to Access the Support Tools Dashboard](#)
- [About the Support Tools Dashboard](#)
- [About Support Tools Utilities](#)
- [How to Use the System Management Screen](#)
- [About Support Tools Security](#)

Part 4: Using Support Tools Utilities

Section I. Using Cisco Web Utilities

About Cisco Web Utilities

Cisco Web utilities are a set of Cisco-authored utilities specifically designed for use with the Support Tools 1.0(1). They are installed to all Support Tools nodes.

All Cisco Web utilities can be run either from the Support Tools Dashboard or via command line on an individual node. Command-line access to individual nodes can be local or remote (via methods like TelNet, pcAnywhere, etc.). However, they were primarily designed for Web access through the Dashboard.

All Cisco Web utilities allow you to save (to the Repository) and download file images of the data they collect.

Note: Within the Dashboard, the functionality of some Cisco Web utilities is distributed among several different screens (pages). For example, Processes and Services utility functionality is divided among a Processes screen and a Services screen.

Cisco Web Utilities at a Glance

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	External GUI - No Dashboard or Command-Line Access:
Log Collector	Use to collect and view log files from ICM nodes. Allows you to create merged logs from two ICM nodes for comparative analysis.	All Support Tools nodes		
Registry Compare and Repair	Use to view, compare and repair ICM node registry entries.	All Support Tools nodes	X	
System Interrogate	Use to display system information (for example, OS, hardware, ICM, SQL).	All Support Tools nodes		
Processes and Services	Use to view and stop Processes and services running on an ICM node.	All Support Tools nodes	X	

Section II. Using Cisco Web Utilities from the Dashboard

How to Use the System Interrogate Screen

Use the *System Interrogate screen* to view information about the current system. You can:

- View system information on:
 - i. Operating system and hardware
 - ii. ICM node
 - iii. Third-party products
- Save current system information to a file

To open the System Interrogate screen, from the Dashboard menu, select Cisco Web Tools > System Interrogate.

To View System Information

To view operating system and hardware information for the current system:

1. Click the Interrogate System button. The screen displays:

OS/Hardware Information

Field:	Description:
OS Type	The operating system and the partition on which the OS is installed.
OS SP Level	OS Service Pack number.
OS Version	OS build number and service pack.
CPU Info	Processor type and speed in Mhz.
Total Physical Memory	Amount of physical memory installed on the system.
Free Physical Memory	Amount of physical memory not in use at the time of the query.
Total Virtual Memory	Total amount of virtual memory the system is configured to support.
Free Virtual Memory	Amount of virtual memory available at the time of the query.
Disk Space	Disk space available and used on all local disk drives.
Page File Size	Maximum size of the paging file.

ICM Information

Field:	Description:
ICM Version	Major version number of the ICM product installed on the System.
Build Number	Build number of the currently installed ICM product.
ICM NODE	Unique identifier for an ICM node (for example, PG2A). The ICM node types include: <ul style="list-style-type: none"> • Router • Logger • PG • AW • Distributor
Engineering Special (Patch) Level	Indicates how many Engineering Special (patches) have been installed on the system.
Patches Applied	Actual patches applied to the system can fall into two categories: <ul style="list-style-type: none"> • Limited Release Patches (LRPs) • RollUp Patches (RUPs)
Install Drive	Drive on which ICM product is installed.
Install Side	In a duplexed system, the side of the ICM that was last installed.
Customer Name(s)	Customer name, or names, for a CICM.
Current Instance	Defines which customer is selected for processing by the AW.
MDS Buffer	ICM Message Delivery System component information. Includes: <ul style="list-style-type: none"> • Buffer Limit • Buffer Max Free • Clients
Domain Controller	The Domain Controller or Active Directory server.
Version/Patch Info	Version/patch information for all ICM components installed.

Third-Party Information

Field:	Description:
SQL Server	ICM database information. Includes: <ul style="list-style-type: none">• SQL Service Pack Version• SQL Config Settings• SQL Database Memory Available• SQL Database Memory In Use• SQL Disk Space Used• SQL Disk Space Free
Internet Information Server (IIS)	Information on the IIS Web server. Includes: <ul style="list-style-type: none">• IIS Settings• Version
ODBC Drivers	Information on Open Data Base Connectivity drivers utilized by various ICM components. Includes: <ul style="list-style-type: none">• ODBC driver versions• Service Pack Number• Hot Fixes
JTAPI	Information on the Java Telephony API component of the IPCC solution. Includes: Version

To Save System Information to a File

As desired, you can save current system information to a file in the Support Tools Repository (for example, as a backup before editing).

To save system information to a file:

1. If desired, click the Refresh button to ensure that the latest system data is displayed on the screen.
2. Click the Save button. A Save dialog box opens, displaying a default name for the file (Sysquery_<Hostname>_<Datetime>.xml).
3. Accept the default name or enter an alternate. Note that:
 - Files can only be saved to the Repository. Entering a path while saving has no effect.

- All files are saved to an XML-formatted text file. While you can enter any extension you like, this will not affect the format in which the file is saved.

4. Click OK. The file is saved to the Repository on the Support Tools Server.

Subsequent to saving, you can view or download saved files by selecting System interrogate > Files from the Dashboard menu.

See Also

For related information, see:

[About Cisco Web Utilities](#)

[How to Configure SQL for Use with the System Interrogate Tool](#)

How to Use the Registry Screen

Use the *Registry screen* to:

- View the ICM Windows registry settings for the current system
- Save registry settings to a file

To open the Registry screen, from the Dashboard menu, select Cisco Web Tools > Registry.

Note: This utility is available to privileged users only. It will not appear on the Dashboard menu for non-privileged users.

To View an ICM Registry

The Registry screen automatically displays the ICM windows registry settings for the current system. Only ICM registry settings can be viewed through the screen; registry settings for other applications are not visible.

To view an ICM registry:

1. Click the Get Registry button. The Cisco branch of the current system's registry displays.
2. Expand any key branch by clicking its right arrow. View any key by clicking its name.

Note: If you suspect the system's registry settings have changed since you first connected to it, click the Refresh button to refresh the view of the current registry.

Viewing Registries for Multiple Customer Instances

For machines with multiple customer instances, the Registry screen only retrieves and displays the registry settings for one instance at a time. By default, the screen retrieves the registry for the first customer instance it encounters based on an alphabetical listing of instance names.

To retrieve the registry for a different instance:

1. In the registry tree, navigate to and expand the ICM branch. All customer instances are displayed. The currently loaded instance will display an arrow to the left of its name, indicating content.
2. Click the name of the instance whose registry you want to retrieve. A message

displays stating: "Data for this key has not been fetched."

3. Click the Fetch Registry link to retrieve registry data for that instance.

To Save Registry Settings to a File

As desired, you can save current registry settings to a file in the Support Tools repository (for example, as a backup before editing). Note that:

- Support Tools automatically downloads registry files at the time you connect to a system. If you suspect the system's registry settings have changed since you first connected to it, click the Refresh button to download the current registry.
- Saving saves the complete Cisco branch of your registry tree--not just the currently expanded portion of that branch.

To save registry settings to a file:

1. Click the Save button. A Save dialog box opens, displaying a default name for the file (Registry<Hostname><Datetime>.xml).
2. Accept the default name or enter an alternate. Note that:
 - Files can only be saved to the Repository. Entering a path while saving has no effect.
 - All files are saved to an XML-formatted text file. While you can enter any extension you like, this will not affect the format in which the file is saved.
3. Click OK. The file is saved to the Repository on the Support Tools Server.

Subsequent to saving, you can view or download saved files by selecting Registry > Files from the Dashboard menu.

See Also

For related information, see:

About Cisco Web Utilities

How to Use the Files Screen

How to Use the Compare Registries Screen

For more information on specific ICM registry keys and settings, see:

Cisco TAC ICM Technical Notes on Cisco.com

Your ICM (Enterprise, IPCC, and Hosted) Technical Documentation

How to Use the Files Screen

Use the *Files screen* to work with Support Tools files previously saved to the Support Tools Repository. You can:

- View saved files
- Rename saved files
- Delete saved files
- Download saved files

You can access the following saved file types through the screen:

- System Interrogate files
- Registry files
- Registry Comparison files
- Process files
- Services files

To open the Files screen, from the Dashboard menu, select Cisco Web Tools > <file_type> > Files. For example to view saved registry files, select Registry > Files. The Files screen automatically displays all of the files of that type currently stored in the Repository.

Note: This utility is available to privileged users only. It will not appear on the Dashboard menu for non-privileged users with the following exception: non-privileged users *can* view System Interrogate files.

To View a Saved File

To view a saved file:

1. Check the file's Select check box.
2. Click the View button. The contents of the file displays.
3. To return to the file list, on the Dashboard menu, select <file_type> > Files.

To Rename a File

To rename a saved file:

1. Check the file's Select check box.
2. Click the Rename button. A Save dialog opens, displaying the current name of the file.
3. Modify the name as desired. Note that all files are saved to an XML-formatted text file. While you can enter any extension you like, this will not affect the format in which the file is saved.

To Delete a File

To delete a file from the Repository:

1. Check the file's Select check box.
2. Click the Delete button.

To Download a File

As desired, you can download a copy of a file from the Repository to your local computer.

To download a file from the Repository:

1. Check the file's Select check box.
2. Click the Download button. A popup window opens containing two links: Download Text and Download XML.
3. To download a file:
 - As plain text with no XML formatting, right-click the Download Text link and select Save Target As from the popup menu. A Save As dialog box opens.
 - With any XML formatting intact, right-click the Download XML link and select Save Target As from the popup menu. A Save As dialog box opens.
4. Specify the location where you want to save the download.
5. Click Save.

How to Use the Compare Registries Screen

Use the *Compare Registries* screen to:

- Compare the ICM Windows registry settings between:
 - i. The current system and another system
 - ii. The current system and a registry file stored in the Support Tools repository
 - iii. Two registry files in the repository
- Copy values from one system to another, or from a file to the current system
- Save a comparison to a file in the Repository

To open the Compare Registries screen, from the Dashboard menu, select Cisco Web Tools > Compare Registries.

Note: This utility is available to privileged users only. It will not appear on the Dashboard menu for non-privileged users.

To Compare the Current System to Another

To compare registry settings from the current system to those of another system:

1. Select the Compare to Another System radio button.
2. Click the Compare Registries button. The systems list displays, showing all nodes defined in Support Tools.
3. Highlight the system you want to use.
4. Click the Compare Registries button.

To Compare the Current System to a Saved Registry File

To compare registry settings from the current system to those of a registry file previously saved in the Support Tools Repository:

1. Select the Compare to Saved Registry File radio button.
2. Click the Compare Registries button. A list of registry files saved in the Repository displays.

3. Select the file you want to compare to.
4. Click the Compare Registry to File button.

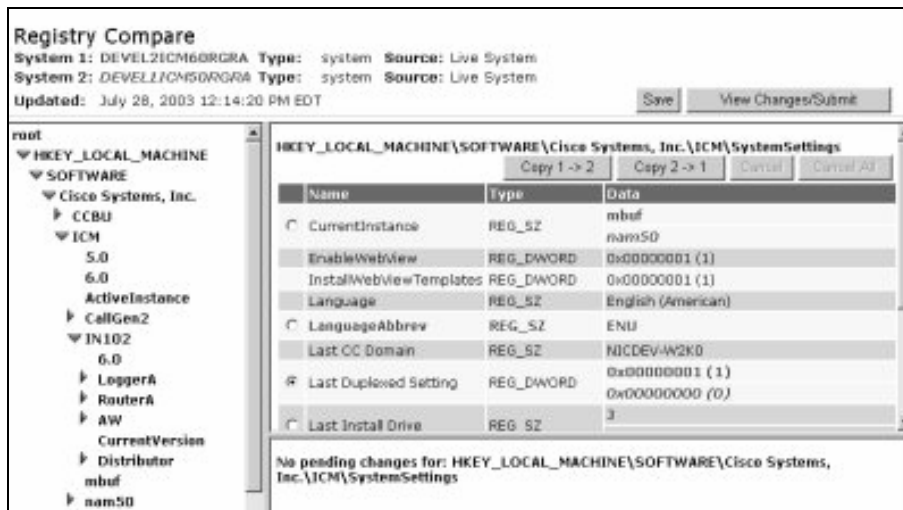
To Compare Two Saved Registry Files

To compare the contents of registry settings from two registry files previously saved to the Support Tools Repository:

1. Select the Compare Two Saved Registry Files radio button.
2. Click the Compare Registries button. A list of registry files saved in the Repository displays.
3. Select the files you want to compare.
4. Click the Compare Registries Files button.

Understanding the Compare Registries Display

Once you have selected the systems or files you want to compare, the Compare Registries screen displays the comparison in a three-pane view as shown in this example:



- A. The top pane shows the systems or files being compared. System 1, the current system or first file being compared, is color-coded in **brown bold**. System 2 is color-coded in **green italic bold**.
- B. The left-hand pane shows a tree view of keys which includes keys from both systems.

- C. The right-hand pane shows (at the top) the path for the selected key, and (beneath) values for that key. Each value is composed of three parts: a name, a type, and data. These are color coded as follows:
- Parts of a value that are identical on both systems are shown in black normal font.
 - Parts of a value that exist only on System 1 are shown in **brown bold**.
 - Parts of a value that exist only on System 2 are shown in *green italic bold*.

Note: For information on how to copy key values from one system to another, see [To Copy Keys Between Registries and Files](#) below.

Viewing Registry Keys for Multiple Customer Instances

For machines with multiple customer instances, the Compare Registries screen only retrieves and displays the registry settings for one instance at a time. By default, the screen retrieves the registry for the first customer instance it encounters on each machine based on an alphabetical listing of instance names. (The currently loaded instance will display an arrow to the left of its name, indicating content.)

To retrieve the registry for a different instance:

1. In the registry tree, navigate to and expand the ICM branch. All customer instances are displayed. The currently loaded instance will display an arrow to the left of its name, indicating content.
2. Click the name of the instance whose registry you want to retrieve. A message displays stating: "Insufficient data available for comparison."
3. Click the Compare Registries Again link to retrieve registry data for that instance.

To Copy Key Values Between Registries and Files

After initiating a registry comparison, you can copy specific key values from one system to another in order to synchronize similar systems, or to correct missing or incorrect values on a system.

You can copy key values from one system to another or from a file to the current system. Note that you cannot copy values between files, or from a system to a file.

Caution: Incorrect modification of registry settings can have serious consequences. Do not change ICM registry settings unless it is absolutely necessary and you are fully aware of the ramifications. It is strongly advised that you first save a registry comparison to a file before making changes.

To copy registry key values:

1. With a comparison displayed, navigate the registry tree to the key whose value

you want to copy.

2. Select the value by clicking its radio button. Values whose data cannot be copied (for example, because the values are already identical, or because you are comparing two files) will not have a radio button.

Note: For an explanation of key value color-coding, see Understanding the Compare Registries Display above.

3. Click the appropriate Copy button (Copy 1 > 2 or Copy 2 >1) to indicate the direction in which the value should be copied. If you are comparing a system to a file, you can only copy to the system.
4. Continue selecting additional values, as desired. Values selected for copying are listed in the lower-right portion of the screen. As necessary, to undo:
 - The most recent selection, click the Cancel button
 - All selections, click the Cancel All button
5. To proceed with changes, click the View Changes/Submit button. All pending changes are listed.
6. Click Submit to finalize your changes.

To Save a Registry Comparison to a File

As desired, you can save a registry comparison to a file in the Support Tools repository for future study or reference. Saving saves the complete retrieved Cisco branch of both registry trees--not just the currently expanded portion of that branch.

Note that in the case of machines with multiple customer instances, only values from the currently retrieved instances are included in the file.

To save a registry comparison to a file:

1. Click the Save button. A Save dialog box opens, displaying a default name for the file (RegCompare_<system1>_<system2>_<Datetime>.xml)...
2. Accept the default name or enter an alternate. Note that:
 - Files can only be saved to the Repository. Entering a path while saving has no effect.
 - All files are saved to an XML-formatted text file. While you can enter any extension you like, this will not affect the format in which the file is saved.
3. Click OK. The file is saved to the Repository on the Support Tools Server.

Subsequent to saving, you can view or download saved files by selecting Compare Registries > Files from the Dashboard menu.

See Also

For related information, see:

[About Cisco Web Utilities](#)

[How to Use the Files Screen](#)

[How to Use the Registry Screen](#)

For more information on specific ICM registry keys and settings, see:

[Cisco TAC ICM Technical Notes on Cisco.com](#)

[Your ICM \(Enterprise, IPCC, and Hosted\) Technical Documentation](#)

How to Use the Processes Screen

Use the *Processes screen* to:

- View all processes running on the current system
- Terminate processes on the current system
- Save a list of processes to a file

To open the Processes screen, from the Dashboard menu, select Cisco Web Tools > Processes.

Note: This utility is available to privileged users only. It will not appear on the Dashboard menu for non-privileged users.

To View Processes

To view processes for the current system:

1. Click the View Processes button. The screen displays all processes currently running on the system. For each process, the screen displays:
 - Name: The name of the process.
 - PID: The process's ID number.
 - Type: Either Critical (blue), Known, (green), or unknown (red). This categorization for each process is based on modifiable process definitions stored in the Processes Definition List.
 - Information: Additional information about the process as defined in the Processes Definition List.

Note: Support Tools automatically downloads the list of current processes at the time you connect to a system. If you suspect the system's processes have changed since you first connected to it, click the Refresh button to refresh the list.

To Terminate a Process

To terminate a process on the current system:

2. With the list of system processes display, select the check box for one or more processes you want to stop.

Note: You cannot stop Critical (blue) processes. Critical processes are those that your company has designated as essential to the health of the current system. For these processes, the Select/Stop check box is disabled.

3. Click the Terminate button. A confirmation screen displays, listing the processes selected for termination.
4. Click the Terminate button.

To Save the Processes List to a File

As desired, you can save a system's current processes list to a file in the Support Tools repository. These files can later be viewed or downloaded locally.

To save processes to a file:

1. Click the Save button. A Save dialog box opens, displaying a default name for the file (<Listtype><Hostname><Datetime>.xml).
2. Accept the default name or enter an alternate. Note that:
 - Files can only be saved to the Repository. Entering a path while saving has no effect.
 - All files are saved to an XML-formatted text file. While you can enter any extension you like, this will not affect the format in which the file is saved.
3. Click OK. The file is saved to the Repository on the Support Tools Server.

Subsequent to saving, you can view or download saved files by selecting Processes > Files from the Dashboard menu.

See Also

For related information, see:

How to Modify the Process Information List

How to Use the Services Screen

Use the *Services screen* to:

- View all Services installed on the current system
- Stop and start services on the current system
- Save a list of Services to a file

To open the Services screen, from the Dashboard menu, select Cisco Web Tools > Services.

Note: This utility is available to privileged users only. It will not appear on the Dashboard menu for non-privileged users.

To View Services

To view services for the current system:

1. Click the View Services button. The screen displays all services on the system. For each service, the screen displays:
 - A check box whose column (Start/Stop) indicates whether the service is currently running.
 - Name: The name of the service.
 - Type: Either Critical (blue), Known, (green), or unknown (red). (Service type is defined in the user-modifiable `processinfo.xml` file.)
 - Information: Additional information about the service as defined in the Services Definition List.

Note: Support Tools automatically determines the state of current services (started or stopped) at the time you connect to a system. If you suspect these states have changed since you first connected to the system, click the Refresh button to refresh the list.

To Stop or Start a Service

To stop or start a service on the current system:

1. With the list of services displayed, select the appropriate check box (Start or Stop) for one or more services.

Note: You cannot stop Critical (blue) services. Critical services are those that your company has designated as essential to the health of the current system. For these services, the Select/Stop check box is disabled.

2. Click the Start/Stop Services button. A confirmation screen displays, listing the services selected for an action.
3. Click the Start/Stop Services button.

Caution: Prior to stopping a service, you should be aware of the significance of stopping it, including any dependant services that may be affected. Note that the Cisco CCBU Support Tools Node Agent service is required for the Support Tools Server to maintain a connection with the current system. Stopping this service will terminate your connection.

Stopping and Starting Dependent Services

The Services screen will not allow you to stop services with peer or child dependencies while those dependencies are running. If you attempt to stop such a service, the Dashboard returns a message alerting you that dependencies exist.

To Save the Services List to a File

As desired, you can save a system's Services list to a file in the Support Tools repository. These files can later be viewed or downloaded locally using the Files screen.

To save Services to a file:

1. Click the Save button. A Save dialog box opens, displaying a default name for the file (Services<Hostname><Datetime>.xml).
2. Accept the default name or enter an alternate. Note that:
 - Files can only be saved to the Repository. Entering a path while saving has no effect.
 - All files are saved to an XML-formatted text file. While you can enter any extension you like, this will not affect the format in which the file is saved.

3. Click OK. The file is saved to the Repository on the Support Tools Server.

Subsequent to saving, you can view or download saved files by selecting Services > Files from the Dashboard menu.

See Also

For related information, see:

[How to Modify the Process Information List](#)

How to Use the Log Collector

Use the Log Collector to collect logs from ICM or related Cisco product nodes running the Support Tools Node. Multiple collected logs can be retrieved as separate files or merged into a single file to provide a consolidated unified time-sorted view.

Products Supported for Log Collection

Support Tools 1.0(1) allows you to collect logs for:

- ICM (AWs, PGs, Call Routers, and Loggers)
- Call Manager
- Collaboration Server
- E-Mail Manager
- Media Blender

Note: Support Tools 1.0(1) does not support log collection for CTIOS.

Log Collection: General Steps

In order to collect logs in Support Tools you must:

1. Create log groups: A log group defines the products, systems, and processes whose logs you want to collect. You must create a log group before you can begin collecting logs. You can create different log groups to target different products and systems.

Log group definitions are saved in the Support Tools Repository and are available for subsequent re-use by other Support Tools users. They can be modified or deleted as necessary.

2. Create a log collection request: A log collection request defines the time range of logs to collect, what log groups to include in the request, and whether the logs should be merged into a single file, or delivered as individual files.

Each log collection request can be run only once. The logs it collects are stored in the Repository where they remain until you delete them (by deleting the log collection to which they belong). Collected logs can be downloaded locally for viewing.

What are Merged Logs?

The Log Collector allows you to create *merged logs*, logs which combine different logs from different systems into a single file. Within a merged log, individual log entries are listed by date and time, allowing you to track, for example, the routing of an individual call across different ICM systems.

There is no limit to the number of individual system logs that can be included in a single merged log file. Note however that larger logs will take longer to collect.

When you download a merged log collection, you can download both the individual associated logs in addition to the merged log itself.

Merge Log Limitations

Merged logs are subject to the following limitations:

- To use merged log capabilities, each Support Tools Server and node must be able to communicate with the same Simple Network Time Protocol (SNTP or NTP) server. *If any system in a merged log collection cannot communicate with the SNTP server, the merge will fail, and the outputted log will contain the alert: "Missing SNTP data for all of the following files logs prevented merging logs:..."*

For more information, see *How to Collect Information for a Support Tools Installation*.

- Some logs do not record time in milliseconds: Some logs record events up to the second, while others record seconds and milliseconds. The Merge logic assumes zeros when milliseconds are missing.
- Even with a properly configured SNTP server, there can be slight drift in time: Used in conjunction with a properly configured SNTP server, each Support Tools node tracks deviation between the system clock and the network standard clock. That difference is used to adjust each log entry. Since time is adjusted in increments of 8 minutes, minor millisecond differences can occur.
- Within a merged log, event ordering can be altered by OS preemption: Within the same system, processes are pre-empted by the OS. It is possible for ordering in the log to be affected by OS preemption.

Viewing Entries in Merged Logs

Log entries in merged logs are prefaced by the following identifiers:

ICM Log Entries	ICM @ <host_name> + <customer_instance>, <ICM_node>, <process>: (<utc date/time>) <local date/time>
CCM Log Entries	CCM @ <host_name> + <CCM_node>: >: (<utc date/time>) <local date/time>

CCS, CEM, and CMB Log Entries	<product> @ <host_name> +: >: (<utc date/time>) <local date/time>
-------------------------------	---

For example, an ICM log entry might be prefaced by:

```
ICM@my_logger + instance_1, logger_a, ccag: (09/04/2003 18:30:00)
09/04/2003 12:30:00
```

Note that in a merged log:

- UTC date/time represents the adjusted date/time of a log entry based on your network's SNTP server clock.
- Local date/time represents the unadjusted date/time of a log entry based on the clock of the system from which the log was derived.

See Also

For related information, see:

[How to Use the Create Log Group Screen](#)

[How to Use the Log Groups Screen](#)

[How to Use the Create Log Collection Screen](#)

[How to Use the Log Collections Screen](#)

How to Use the Create Log Group Screen

Use the *Create Log Group Screen* to create a log group. A log group is a specification that defines the products, systems, and processes you want to pull logs for.

To open the Create Log Group screen, from the Dashboard menu, select Cisco Web Tools > Log Collector > Create Log Group.

To Create a Log Group

To create a log group:

1. Select the Cisco product(s) whose logs will be collected by this group. You can collect logs for:
 - ICM (AWs, PGs, Call Routers, and Loggers)
 - Call Manager
 - Collaboration Server
 - E-Mail Manager
 - Media Blender
2. Click Next. All nodes defined on the Support Tools system list that belong to the selected products are listed.

Note: This list may also include any nodes whose product type is unknown (blank). The most common reason for an unknown product type is that the node has never been contact by the Support Tools Server. For example, it was added to the system list (via System Management screen) but the connection was not tested or the node has never been selected as a current system.
3. Select the system(s) whose logs you want to collect.
4. Click Next.
5. Enter a name (up to 32 characters long), and optionally a description, for the log group. It's a good idea to choose an intuitive name that will help other users understand what type of logs this group collects.
6. If the product you selected is:
 - ICM, you must indicate which ICM system processes you want to collect logs for. For each ICM system, the screen lists:

- System: The ICM system name
- Customer: The ICM customer instance(s) on the system
- Node: The ICM node ID (for example, pg1a, pg1b, router a, router b)
- Process: The process(es) on that node for which logs can be collected.

CCM, you must indicate which CCM processes you want to collect logs for. For each CCM system the screen displays:

- System: The CCM system name
- Node: The CCM process(es) for which logs can be collected--CCM or CTI.

7. If any of the systems you selected are unavailable for log collection (for example, if they are offline, a message will display on the screen alerting you to the fact.
8. Click Finish. The log group is now available to use in a log collection (on the Create Log Collection screen).

See Also

For related information, see:

[How to Use the Log Collector](#)

[How to Use the Log Groups Screen](#)

[How to Use the Create Log Collection Screen](#)

[How to Use the Log Collections Screen](#)

How to Use the Log Groups Screen

Use the *Log Groups Screen* to administer log groups once they have been created. You can:

- View details of log groups
- Edit log group definitions
- Rename log groups
- Delete log groups

To open the Log Groups screen, from the Dashboard menu, select Cisco Web Tools > Log Collector > Log Group.

To View Details of a Log Group

To view details of a log group:

1. Check the file's Select check box.
2. Click the View button. For ICM systems, the screen displays:
 - System: The ICM system name
 - Customer: The ICM customer instance(s) on the system
 - Node: The ICM node name (for example, pg1a, pg1b, router a, router b)
 - Process: The process(es) on that node for which logs can be collected.

For CCM systems the screen displays:

- System: The CCM system name
- Node: The process(es) for which logs can be collected--CCM or CTI.

For other systems (CCS, CMB, and CEM) the screen displays:

- System: The system name
- Product: The product type: CEM, CCS, CMB

To Edit a Log Group Definition

Once created, log groups do not update automatically to reflect subsequent changes made to products installed on the systems they include; If and when these changes occur, you must edit the group to correct its definition. Failure to do this will prevent the successful log collection using that group.

To edit a log group's definition:

1. Check the group's Select check box.
2. Click the Edit button.
3. Select the Cisco product(s) whose logs will be collected by this group. You can collect logs for:
 - ICM (AWs, PGs, Call Routers, and Loggers)
 - Call Manager
 - Collaboration Server
 - E-Mail Manager
 - Media Blender
4. Click Next. All nodes defined on the Support Tools system list that belong to the selected products are listed.
5. Select the system(s) whose logs you want to collect.
6. Click Next.
7. Optionally, edit the description for the log group.
8. If the product you selected is:
 - ICM, you must indicate which ICM system processes you want to collect logs for. For each ICM system, the screen lists:
 - System: The ICM system name
 - Customer: The ICM customer instance(s) on the system
 - Node: The ICM node ID (for example, pg1a, pg1b, router a, router b)
 - Process: The process(es) on that node for which logs can be collected.
 - CCM, you must indicate which CCM system processes you want to collect logs for. For each CCM system the screen displays:
 - System: The CCM system name.

- Node: The CCM process(es) for which logs can be collected--CCM or CTI.
9. Click Finish. The log group is now available to use in a log collection (on the Create Log Collection screen).

To Rename a Log Group

To rename a log group:

1. Check the group's Select check box.
2. Click the Rename button. A save dialog opens, displaying the current name of the file.
3. Modify the name as desired. Maximum length for name is 32 characters.
4. Click Save.

To Delete a Log Group

To delete a log group:

1. Check the group's Select check box.
2. Click the Delete button.
3. A message displays prompting you to confirm the deletion.
4. Click OK.

See Also

For related information, see:

[How to Use the Log Collector](#)

[How to Use the Create Log Group Screen](#)

[How to Use the Create Log Collection Screen](#)

[How to Use the Log Collections Screen](#)

How to Use the Create Log Collection Screen

Use the *Create Log Collection Screen* to create a log collection. A log collection is a specification that defines the log group(s) and time range you want to collect logs for.

To open the Create Log Collection screen, from the Dashboard menu, select Cisco Web Tools > Log Collector > Create Log Collection.

To Create a Log Collection

To create a log group:

1. Enter a name (up to 32 characters long) for the log collection. It's a good idea to choose an intuitive name that conveys the attributes of the log collection.
2. Specify a time range of logs to collect: All or Range. If you select Range, specify the start and end period time.
3. If desired, check Merged Log to create a single merged log file from the logs collected by this group.

Note: When you later download a merged log, the collection will include the individual associated log files in addition to the merged log itself.

4. If desired, for ICM system logs only, check Include Binary Data. This causes the log file to include binary data such as memory allocation, buffer size, etc., but may also substantially increase file size.
5. Select one or more log groups to collect logs from.
6. Click the Start Log Collection button. The log collection process begins. The amount of time this process takes to complete depends on the number and size of the logs you requested. You can view collected logs or check the status of requests on the Log Collections screen.

Note: It is not necessary to remain in-session while a log collection runs. Exiting your Dashboard session will not terminate a log collection. You can proceed to use other Dashboard while a Log Collection runs.

See Also

For related information, see:

[How to Use the Log Collector](#)

[How to Use the Create Log Group Screen](#)

[How to Use the Log Groups Screen](#)

[How to Use the Log Collections Screen](#)

How to Use the Log Collections Screen

Use the *Log Collections Screen* to view and administer collected logs. You can:

- View details of log collections
- Download collected logs
- Delete log collections

To open the Log Collections screen, from the Dashboard menu, select Cisco Web Tools > Log Collector > Log Collections.

To View Details of Log Collections

The Log Collection screen automatically lists each collection currently stored in the Support Tools repository. For each collection, the screen displays:

- Name: The name of the collection.
- Requested: The date/time its request was sent.
- Status: The status of the collection--Ready (complete) or In Progress (incomplete).
- Size: The size of the log files retrieved by the collection.

To Download Collected Logs

Collected logs can only be viewed after downloading to your local machine; they cannot be viewed from within Support Tools.

To download collected logs:

1. Check the Select check box for the collection whose logs you want to view. You can only download the logs from one collection at a time. If necessary, before proceeding, verify that the collection's status is Ready (complete).
2. Click the View button. Details of the collection, along with the individual log files it contains, are displayed. If the log collection included a merged log, both the individual associated log files and the merged log file are listed.
3. Check the Select check box for the logs you want to download. If you select individual logs from a merged log collection, the merged log will automatically download along with the individual logs you select.

4. Click the Download button.
5. When prompted, choose the Save to Disk option.

Note: Logs download in a single zip file which must be extracted in order to view.

Log File Naming Conventions

Log files use the following naming conventions:

- For ICM: <collection-group>{<product>{<NodeAgentHost>(<customer>, <node>, <process>}.log
- For CCM: <collection-group>{<product>{<NodeAgentHost>(<node>}.log
- For other products: <collection-group>{<product>{<NodeAgentHost>}.log

To Delete a Log Collection

To delete a log collection:

1. Check the collection's Select check box.
2. Click the Delete button.
3. A message displays prompting you to confirm the deletion.
4. Click OK.

See Also

For related information, see:

How to Use the Log Collector

How to Use the Create Log Group Screen

How to Use the Log Groups Screen

How to Use the Create Log Collection Screen

Section III. Using Cisco Web Utilities from a Command Line

About Using Cisco Web Utilities from a Command Line

While primarily designed for use through the Dashboard, Cisco Web Utilities can also be run from a command line on any Support Tools node. This can be useful when access to the Support Tools Server/Dashboard is not available due to a down machine or connection.

The same functionality available in the Dashboard versions of these utilities is available in their command line counterparts. As GUIs are generally easier to learn, it is a good idea to familiarize yourself with these tools in the Dashboard before attempting to use them from a command line.

Web Utilities Location

Cisco Web utilities are installed and can be run from any Support Tools node. Their default location (modifiable during installation) is: `c:\cisco\ccbu\supporttools`.

Command-Line Mode vs Interactive Mode

When run from a command line, Cisco Web utilities can be run in either *command-line mode* or *interactive mode*:

- In command-line mode, you enter the command invoking the utility, along with specific arguments, on a single line. After the command executes the utility automatically shuts down.

Example: `c:\cisco\ccbu\supporttools> interrogate /system logger_a /query`

- In interactive mode, you enter the command invoking the utility on a separate command line. After that, you can enter as many successive commands as desired. The utility does not terminate until you explicitly quit.

```
c:\cisco\ccbu\supporttools> interrogate

> system logger_a

> query

> quit
```

Note: The Log Group and Log Collection utilities are only available in interactive mode; they cannot be used in command-line mode.

Selecting a System to Use

When run from a command line, unless otherwise specified, Cisco Web utilities assume that the local machine is both:

- A) The *application server* (the machine from which the utility should be run), and
- B) The *target system* (the machine to be queried by the utility).

For example, to run the System Interrogate utility on the local machine, and to target its query at that same machine, enter:

```
>interrogate /query
```

Selecting a Different Target System

If desired, you can use the `/system` option to specify a different machine as the target system of a command. The `/system` option can only be used when the application server is a Support Tools Server; You cannot target a remote system when using a node as your application server--node can only target themselves.

For example, to target a System Interrogate query at a machine named `logger_b`, enter:

```
>interrogate
>system logger_a
>query
```

In the example above, the System Interrogate utility is being run locally (from a Support Tools Server) and targeted at a node named `logger_b`.

Note: This command is not available for the Log Group and Log Collection utilities. In those utilities, the target system specification is built into the log group(s) being used in the request.

Selecting a Different Application Server

If desired, you can run a utility remotely from a Support Tools Server. For example, to run the System Interrogate utility from a Support Tools Server named `appserver_a`, enter:

```
>interrogate st_server_a
>system logger_a
>query
```

In the example above, the System Interrogate utility is being run remotely from a Support Tools Server named `st_server_a` and targeted at a node named `logger_b`.

If a remote Support Tools Server is to serve as both the application server and the target of the command, you must specify that machine as both the application server and the target system. For example:

```
>interrogate st_server_a  
  
>system st_server_a  
  
>query
```

Viewing a list of targetable systems

Once connected to an application server, you can use the `systemlist` (or `syslist`) command to view a list of all systems that can be selected for targeting. Use the `/check` option with `systemlist` to verify connection status to available systems.

For example, entering `systemlist /check` returns output such as:

Hostname:	IP Address:	System Type:	Connection:
10.86.128.84	10.86.128.84	ICM	c
10.86.142.23	10.86.142.23	CEM	c
10.86.142.39	10.86.142.39	ICM	X

A lower-case c indicates a live connection. An X indicates a failed connection.

Saving, Viewing, and Retrieving Files

Most Cisco Web utilities allow you to save, view, and download files from the Support Tools Repository. Each Support Tools node has its own local Repository. When using Web utilities from a command line on a node, the Repository in use is the Repository on that node (or whatever machine has been specified as the application server).

Getting Help for Command Line Tools

Use the `cmdhelp` or `chelp` command to view a complete list of options for any tool.

To view usage for a specific command, enter: `<command_name> /?`.

Embedded Spaces

In command-line arguments, remember to place paths, file names, etc. that contain embedded spaces in quotes. Failure to do this will prevent a command from executing correctly.

Example: > view "my file.txt"

How to Use the Services Utility from a Command Line

While primarily designed for use from the Support Tools Dashboard, the *Services* utility can also be run from a command line any node, as desired.

Use the Services utility to:

- View information on all services installed on a system
- Stop or start services on a system
- Save a list of services to a file

To Access the Services Utility from a Command Line

To run the Services utility in interactive mode from a command line:

1. From a command prompt on a Support Tools node, navigate to `<support_tools_root>`.
2. Enter `servicecontrol`. Optionally in the same line, to specify a different machine to serve as the application server, enter its DNS or IP address. If none is specified, the local machine serves as the application server. Press `<Enter>`.
 - To view a list of systems defined in the current system's system list, use the `systemlist` command. To view connection status for systems in the list, use the `systemlist /check` command.
3. Enter commands as described in the Command Line Options section below.

Note: For additional information on basic Web Tools command line use, including system selection options and command-line mode vs interactive mode see [About Using Cisco Web Utilities from a Command Line](#).

Using the Services Utility from a Command Line

When using the Services utility from a command line, note that:

- Unlike in Dashboard use, you can stop services whose Type is Critical (service type is defined in the user-modifiable `processinfo.xml` file.)
- Prior to stopping a service, you should be aware of the significance of stopping it, including any dependant services that may be affected.
- The Services utility will not allow you to stop services with peer or child

dependencies while those dependencies are running. If you attempt to stop such a service, the utility returns a message stating that dependent services are running.

Command Line Options

Command:	Description:	Example:
cmdhelp, chelp	Displays a list of commands specific to this utility. Note: Using Help or ? also displays this list, but includes several additional ICM commands (e.g., echo, error_stop) not used by this utility.	>cmdhelp
/?	Displays syntax for a specified command.	><command> /?
appserver	Specifies the system on which the utility should run. If not specified, the utility is run on the local system.	><app_servername> /<options>
system	Specifies the target system the utility should run against. If not specified, the utility is run against the local system.	> /system <host_name> /<options>
localhost	Sets the network address of name of the target node agent to the local host. Note: By default, unless a different system is specified (using the system command) the local host is assumed to be the target system.	>localhost
list	Displays information on all services on the target system. For each service, this command displays: <ul style="list-style-type: none"> • Service Name: The service's name. • PID: The service's process ID. • Type: Critical, Known, or Unknown, as defined in the target system's processinfo.xml file. • Start (if the process is stopped) or Stop (if the process is started). Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.	>list OR >list <localfile_path\filename>
start	Starts a stopped service on the target system.	>start <pid> <service_name> Note: Enter "0" for the PID when starting a service.
stop	Stops a started service on the target system.	>stop <pid> <service_name>

Command:	Description:	Example:
save	Saves the latest list command results to a services file in the application server's Repository. Saved output is stored as XML-formatted text. If no file name is specified, output is saved to a file named: ServicesList <host_name> <timestamp>.xml. If desired, you can include an argument that specifies a file name.	>save OR >save <filename>
files	Displays the list of services files in the application server's Repository.	>files
view	Displays the contents of a services file from the Repository on the application server. Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.	>view <filename> OR >view <filename> <localfile_path\filename>
remove, rm	Deletes a specified services file from the Repository on the application server.	>remove <filename>
rename, ren	Renames a services file in the Repository on the application server.	>rename <filename> <new_filename>
read_file, read	Directs command input to another input file. For example, you can direct input to run a batch file that contains a series of commands executable by this utility.	>read <filename>
silent	Executes command without displaying output. Note: Available in command-line mode only.	><command> /silent
quit, q	Ends the program.	>quit

Examples

Viewing and Stopping a Service

The following table shows an example of viewing and stopping a service, with a description of each entry.

Command Option:	Description:
>servicecontrol	Invokes the Services utility.
>list	Returns a list of all services installed on the target system, along with their current status.
>stop 01464 "snmp service"	Stops the specified service. Note that due to an embedded space, the service name was placed in quotes.

See Also

For related information, see:

[About Using Cisco Web Utilities from a Command Line](#)

[How to Modify the Process Information List](#)

How to Use the Processes Utility from a Command Line

While primarily designed for use from the Support Tools Dashboard, the *Processes* utility can also be run from a command line any node, as desired.

Use the *Processes* utility to:

- View all processes running on a system
- Terminate processes on a system
- Save a list of processes to a file

To Run the Processes Utility from a Command Line

To run the Processes utility in interactive mode from a command line:

1. From a command prompt on a Support Tools node, navigate to `<support_tools_root>`.
2. Enter `processcontrol`. Optionally in the same line, to specify a different machine to serve as the application server, enter its DNS or IP address. If none is specified, the local machine serves as the application server. Press `<Enter>`.
 - To view a list of systems defined in the current system's system list, use the `systemlist` command. To view connection status for systems in the list, use the `systemlist /check` command.
3. Enter commands as described in the Command Line Options section below.

Note: For additional information on basic Web Tools command line use, including system selection options and command-line mode vs interactive mode see [About Using Cisco Web Utilities from a Command Line](#).

Using the Processes Utility from a Command Line

When using the Processes utility from a command line, note that unlike in Dashboard use, you can stop processes whose Type is Critical (service type is defined in the user-modifiable `processinfo.xml` file.)

Command Line Options

Command:	Description:	Example:
cmdhelp, chelp	Displays a list of commands specific to this utility. Note: Using Help or ? also displays this list, but includes several additional ICM commands (e.g., echo, error_stop) not used by this utility.	>cmdhelp
/?	Displays syntax for a specified command.	><command> /?
appserver	Specifies the system on which the utility should run. If not specified, the utility is run on the local system.	><app_servername> /<options>
system	Specifies the target system the utility should run against. If not specified, the utility is run against the local system.	> /system <host_name> /<options>
localhost	Sets the network address of name of the target node agent to the local host. Note: By default, unless a different system is specified (using the system command) the local host is assumed to be the target system.	>localhost
list	Displays information on all processes on the target system. For each running process, this command displays: <ul style="list-style-type: none"> • Process Name: The process's name. • PID: The process's process ID. • Type: Critical, Known, or Unknown, as defined in the target system's processinfo.xml file. Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.	>list OR >list <localfile_path\filename>
kill	Terminates a started process on the target system.	>stop <pid> <process_name>
save	Saves the latest list command results to a processes file in the application server's Repository. Saved output is stored as XML-formatted text. If no file name is specified, output is saved to a file named: ProcessesList <host_name> <timestamp>.xml. If desired, you can include an argument that specifies a file name.	>save OR >save <filename>
files	Displays the list of processes files in the application server's Repository.	>files

Command:	Description:	Example:
view	Displays the contents of a processes file from the Repository on the application server. Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.	>view <filename> OR >view <filename> <localfile_path\filename>
remove, rm	Deletes a specified processes file from the Repository on the application server.	>remove <filename>
rename, ren	Renames a processes file in the Repository on the application server.	>rename <filename> <new_filename>
read_file, read	Directs command input to another input file. For example, you can direct input to run a batch file that contains a series of commands executable by this utility.	>read <filename>
silent	Executes command without displaying output. Note: Available in command-line mode only.	><command> /silent
quit, q	Ends the program.	>quit

Examples

Viewing and Killing a Process

The following table shows an example of viewing and killing a process, with a description of each entry.

Command Option:	Description:
>processcontrol	Invokes the Processes utility.
>list	Returns a list of all processes running on the target system.
>stop 01464 "snmp service"	Kills the specified process. Note that due to an embedded space, the process name was placed in quotes.

See Also

For related information, see:

[About Using Cisco Web Utilities from a Command Line](#)

[How to Modify the Process Information List](#)

How to Use the System Interrogate Utility from a Command Line

While primarily designed for use from the Support Tools Dashboard, the *System Interrogate* utility can also be run from a command line any node, as desired.

Use the System Interrogate utility to:

- View detailed information about a system's OS, hardware, ICM, and third-party products.
- Save current system information to a file

To Run the System Interrogate Utility from a Command Line

To run the System Interrogate utility in interactive mode from a command line:

1. From a command prompt on a Support Tools node, navigate to `<support_tools_root>`.
2. Enter `interrogate`. Optionally in the same line, to specify a different machine to serve as the application server, enter its DNS or IP address. If none is specified, the local machine serves as the application server. Press `<Enter>`.
 - To view a list of systems defined in the current system's system list, use the `systemlist` command. To view connection status for systems in the list, use the `systemlist /check` command.
3. Enter commands as described in the Command Line Options section below.

Note: For additional information on basic Web Tools command line use, including system selection options and command-line mode vs interactive mode see [About Using Cisco Web Utilities from a Command Line](#).

Using the System Interrogate Utility from a Command Line

Command Line Options

Command:	Description:	Example:
----------	--------------	----------

Command:	Description:	Example:
cmdhelp, chelp	Displays a list of commands specific to this utility. Note: Using Help or ? also displays this list, but includes several additional ICM commands (e.g., echo, error_stop) not used by this utility.	>cmdhelp
/?	Displays syntax for a specified command.	><command> /?
appserver	Specifies the system on which the utility should run. If not specified, the utility is run on the local system.	><app_servername> /<options>
system	Specifies the target system the utility should run against. If not specified, the utility is run against the local system.	> /system <host_name> /<options>
localhost	Sets the network address of name of the target node agent to the local host. Note: By default, unless a different system is specified (using the system command) the local host is assumed to be the target system.	>localhost
query, qry	Displays detailed information about the target system. For a detailed listing of displayed information, see How to Use the System Interrogate Screen. Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.	>list OR >list <localfile_path\filename>
save	Saves the latest query command results to a file in the application server's Repository. Saved output is stored as XML-formatted text. If no file name is specified, output is saved to a file named: SysQuery <host_name> <timestamp>.xml. If desired, you can include an argument that specifies a file name.	>save OR >save <filename>
files	Displays the list of System Interrogate files in the target system's Repository.	>files
view	Displays the contents of a System Interrogate file from the Repository on the application server.	>view <filename> OR >view <filename> <localfile_path\filename>
download, dl	Downloads a local copy of a System Interrogate file from the application server's Repository. Optionally, you can include a /text argument to convert the file from XML-formatted text to plain text.	>download <repository_filename> <local_filename> /text OR >download <repository_filename> <localfile_path\filename> /text
remove, rm	Deletes a specified System Interrogate file from the Repository on the application server.	>remove <filename>

Command:	Description:	Example:
rename, ren	Renames a System Interrogate file in the Repository on the application server.	>rename <filename> <new_filename>
read_file, read	Directs command input to another input file. For example, you can direct input to run a batch file that contains a series of commands executable by this utility.	>read <filename>
silent	Executes command without displaying output. Note: Available in command-line mode only.	><command> /silent
quit, q	Ends the program.	>quit

Examples

Viewing System Information

The following table shows an example of querying system information, with a description of each entry.

Command Option:	Description:
>interrogate	Invokes the System Interrogate utility.
>query	Queries the target system. Returns detailed on that system.

See Also

For related information, see:

About Using Cisco Web Utilities from a Command Line

How to Use the System Interrogate Screen

How to Use the Registry Utility from a Command Line

While primarily designed for use from the Support Tools Dashboard, the *Registry* utility can also be run from a command line any node, as desired.

Use the Registry utility to:

- View the ICM Windows registry settings for the current system
- Save registry settings to a file

To Run the Registry Utility from a Command Line

To run the Registry utility in interactive mode from a command line:

1. From a command prompt on a Support Tools node, navigate to `<support_tools_root>`.
2. Enter `registry`. Optionally in the same line, to specify a different machine to serve as the application server, enter its DNS or IP address. If none is specified, the local machine serves as the application server. Press `<Enter>`.
 - To view a list of systems defined in the current system's system list, use the `systemlist` command. To view connection status for systems in the list, use the `systemlist /check` command.
3. Enter commands as described in the Command Line Options section below.

Note: For additional information on basic Web Tools command line use, including system selection options and command-line mode vs interactive mode see [About Using Cisco Web Utilities from a Command Line](#).

Using the Registry Utility from a Command Line

Command Line Options

Command:	Description:	Example:
cmdhelp, chelp	Displays a list of commands specific to this utility. Note: Using Help or ? also displays this list, but includes several additional ICM commands (e.g., echo, error_stop) not used by this utility.	>cmdhelp
/?	Displays syntax for a specified command.	><command> /?
appserver	Specifies the system on which the utility should run. If not specified, the utility is run on the local system.	><app_servername> /<options>
system	Specifies the target system the utility should run against. If not specified, the utility is run against the local system.	> /system <host_name> /<options>
localhost	Sets the network address of name of the target node agent to the local host. Note: By default, unless a different system is specified (using the system command) the local host is assumed to be the target system.	>localhost
list	Displays the ICM Windows Registry setting for the target system. Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.	>list OR >list <localfile_path\filename>
instance	Used in conjunction with the list command, specifies the customer instance whose registry values will be returned. If not specified, the List command automatically returns registry values for the first customer instance it finds on the target system.	>/list /instance <instance_name>
save	Saves the latest list command results to a registry file in the application server's Repository. Saved output is stored as XML-formatted text. If no file name is specified, output is saved to a file named: Registry <host_name> <timestamp>.xml. If desired, you can include an argument that specifies a file name.	>save OR >save <filename>
files	Displays the list of registry files in the application server's Repository.	>files

Command:	Description:	Example:
view	Displays the contents of a registry file from the Repository on the application server. Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.	>view <filename> OR >view <filename> <localfile_path\filename>
remove, rm	Deletes a specified registry file from the Repository on the application server.	>remove <filename>
rename, ren	Renames a registry file in the Repository on the application server.	>rename <filename> <new_filename>
read_file, read	Directs command input to another input file. For example, you can direct input to run a batch file that contains a series of commands executable by this utility.	>read <filename>
silent	Executes command without displaying output. Note: Available in command-line mode only.	><command> /silent
quit, q	Ends the program.	>quit

Examples

Viewing Registry Information

The following table shows an example of querying registry information, with a description of each entry.

Command Option:	Description:
>registry	Invokes the Registry utility.
>list /instance customer_1	Queries the target system and returns the registry information for a customer instance named customer_1. If a specific instance is not specified, the List command automatically returns registry values for the first customer instance it finds on the target system.
>save	Optionally, saves the data returned to a registry file in the application server's Repository. If no file name is specified, output is saved to a file named: Registry <host_name> <timestamp>.xml. If desired, you can include an argument that specifies a file name.

See Also

For related information, see:

[About Using Cisco Web Utilities from a Command Line](#)

[How to Use the Compare Registries Utility from a Command Line](#)

How to Use the Compare Registries Utility from a Command Line

While primarily designed for use from the Support Tools Dashboard, the *Compare Registries* utility can also be run from a command line any node, as desired.

Use the Compare Registries utility to:

- Compare the ICM Windows registry settings between:
 - i. The current system and another system
 - ii. The current system and a registry file stored in the Support Tools repository
 - iii. Two registry files in the repository
- Copy registry values from one system to another, or from a file to the current system
- Save a comparison to a file in the Repository

To Run the Compare Registries Utility from a Command Line

To run the Compare Registries utility in interactive mode from a command line:

1. From a command prompt on a Support Tools node, navigate to `<support_tools_root>`.
2. Enter `regcompare`. Optionally in the same line, to specify a different machine to serve as the application server, enter its DNS or IP address. If none is specified, the local machine serves as the application server. Press `<Enter>`.
 - To view a list of systems defined in the current system's system list, use the `systemlist` command. To view connection status for systems in the list, use the `systemlist /check` command.
3. Enter commands as described in the Command Line Options section below.

Note: For additional information on basic Web Tools command line use, including system selection options and command-line mode vs interactive mode see About Using Cisco Web Utilities from a Command Line.

Using the Compare Registries Utility from a Command Line

Command Line Options

Command:	Description:	Example:
cmdhelp, chelp	Displays a list of commands specific to this utility. Note: Using Help or ? also displays this list, but includes several additional ICM commands (e.g., echo, error_stop) not used by this utility.	>cmdhelp
/?	Displays syntax for a specified command.	><command> /?
appserver	Specifies the system on which the utility should run. If not specified, the utility is run on the local system.	><app_servername> /<options>
system	Specifies the target system the utility should run against. If not specified, the utility is run against the local system.	> /system <host_name> /<options>
localhost	Sets the network address of name of the target node agent to the local host. Note: By default, unless a different system is specified (using the system command) the local host is assumed to be the target system.	>localhost
compare, comp ./systems ./files ./system2file	Compares the contents of two registries and displays the result. You can compare: <ul style="list-style-type: none"> The current system to a different system Two registry files in the Repository The current system to a registry file in the Repository <p>Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.</p> <p>When viewing results:</p> <ul style="list-style-type: none"> Values whose data are identical in both systems are prefaced by Common Value = Values whose data are unique to the current system are prefaced by SRC = Values whose data are unique to the current system are prefaced by SRC = 	>compare /systems <2nd system> OR >compare /file <reg_file_1> <reg_file_2> OR >compare /system2file <reg_file>
instance	Used in conjunction with the compare command, specifies the customer instance whose registry values will be compared. If not specified, the List command automatically returns registry values for the first customer instance it finds on each compared system.	>compare /systems <2nd system> /instance <instance_name>

Command:	Description:	Example:
apply /src2diff /diff2src	<p>Copies a registry value from one registry to another, or from a file to a registry, using values from the last compare command result.</p> <p>Use the /src2diff and /diff2src arguments to specify which direction the value should be copied.</p> <p>Note that you can only copy values to a registry; you cannot copy a value to a file.</p>	<pre>>apply <keyname> <value> /src2diff OR >apply <keyname> <value> /diff2src</pre>
save	<p>Saves the latest compare command results to a registry file in the application server's Repository.</p> <p>Saved output is stored as XML-formatted text.</p> <p>If no file name is specified, output is saved to a file named: RegCompare <host_name> <timestamp>.xml.</p> <p>If desired, you can include an argument that specifies a file name.</p>	<pre>>save OR >save <filename></pre>
files	Displays the list of RegCompare files in the application server's Repository.	>files
view	<p>Displays the contents of a RegCompare file from the Repository on the application server.</p> <p>Optionally, you can include an argument to dump this output to a local file. Output is stored as XML-formatted text.</p>	<pre>>view <filename> OR >view <filename> <localfile_path>\filename></pre>
remove, rm	Deletes a specified RegCompare file from the Repository on the application server.	>remove <filename>
rename, ren	Renames a RegCompare file in the Repository on the application server.	>rename <filename> <new_filename>
read_file, read	<p>Directs command input to another input file.</p> <p>For example, you can direct input to run a batch file that contains a series of commands executable by this utility.</p>	>read <filename>
silent	<p>Executes command without displaying output.</p> <p>Note: Available in command-line mode only.</p>	><command> /silent
quit, q	Ends the program.	>quit

Examples

Performing a Registry Comparison

The following table shows a sample registry comparison request, with a description of each entry.

Command Option:	Description:
>registry	Invokes the Registry Comparison utility.
>compare /systems logger_b /instance customer_1	Specifies that the local system's registry settings for customer instance_1 be compared to logger_b's settings for that instance. If a specific instance is not specified, the compare command automatically returns registry values for the first customer instance it finds on each system. Note that in some cases these may not be identical instances.
>save	Optionally, saves the data returned to a regcompare file in the application server's Repository. If no file name is specified, output is saved to a file named: Registry <host_name> <timestamp>.xml. If desired, you can include an argument that specifies a file name.

Copying Key Values from one Registry to Another

The following table shows a sample of copying a registry key value from one system to another. Prior to running this command, you must first perform a registry comparison, as shown in the previous example.

Command Option:	Description:
>apply "HKEY_LOCAL_MACHINE\SOFTWARE \Cisco Systems, Inc.\CCBU\Support Tools\Tools\SysQuery" ProgID /src2diff	Specifies that the value ProgID for key HKEY_LOCAL_MACHINE\SOFTWARE\Cisco Systems, Inc.\CCBU\Support Tools\Tools\SysQuery should be copied from the target system to the system specified in the Compare command (in the previous example). Note that because it includes embedded spaces, the key name has been placed in quotes.

See Also

For related information, see:

[About Using Cisco Web Utilities from a Command Line](#)

[How to Use the Registry Utility from a Command Line](#)

How to Use the Log Groups Utility from a Command Line

Use the *Create Log Group Screen* to:

- Create log groups
- View details of log groups
- Edit log group definitions
- Rename log groups
- Delete log groups

To Run the Log Groups Utility from a Command Line

To run the Log Groups utility in interactive mode from a command line:

1. From a command prompt on a Support Tools node, navigate to `<support_tools_root>`.
2. Enter `lgtool`. Optionally in the same line, to specify a different machine to serve as the application server, enter its DNS or IP address. If none is specified, the local machine serves as the application server. Press `<Enter>`.
3. Enter commands as described in the Command Line Options section below.

Note: For additional information on basic Web Tools command line use, including system selection options and command-line mode vs interactive mode see [About Using Cisco Web Utilities from a Command Line](#).

Using the Log Groups Utility from a Command Line

Command Line Options

Command:	Description:	Example:
cmdhelp, chelp	Displays a list of commands specific to this utility. Note: Using Help or ? also displays this list, but includes several additional ICM commands (e.g., echo, error_stop) not used by this utility.	>cmdhelp
/?	Displays syntax for a specified command.	><command> /?
appserver	Specifies the system on which the utility should run. If not specified, the utility is run on the local system.	><app_servername> /<commands>
system	Note: This command is not used with the Log Groups utility.	
localhost	Note: This command is not used with the Log Groups utility.	
create_log_group, clg	Opens a session for creating a log group.	>clg <new_group_name> Note: Maximum length for name is 32 characters.
list_products, lpd	Displays products that can be selected for log collection.	>lpd
list_systems, lsys	Displays systems that can be selected for log collection. For each system for which the information is available, it displays: product system name customer name icm or ccm node name process name	>lsys
add_log, al /product /system /icm_customer /icm_node /icm_process /ccm_node	Adds one log in the current log group. <ul style="list-style-type: none"> For ICM logs you must specify: /product, /system, /icm_customer, /icm_node, and /icm_process. For CCM logs you must specify: /product, /system, and /ccm_node. For CEM, CCM, and CCS logs, you must specify: /product and /system. 	>al /product ICM /system <system_name> /icm_customer <customer_name> /icm_node <node_name> /icm_process <process_name> >al /product CCM /system <system_name> /ccm_node <node_name> >al /product CCS /system <system_name>
remove_log, rl	Removes a log from current log group.	rl <log_name>
list_selected_log, lsl	Displays logs included in the current log group.	>lsl

Command:	Description:	Example:
log_group_description, lgdesc	Optionally, sets a description for the current log collection.	>lgdesc <description>
close_log_group, closelg /save	Closes and saves the current log group to the application server's Repository. When closing a new group, closelg automatically saves the group. When closing an existing group that was opened for editing, you must explicitly choose save by entering: closelg /save YES. To close without saving, enter: closelg /save NO.	>closelg
list_log_groups, lslg	List all log groups in the application server's Repository.	>lslg
display_log_group, dlgl	Displays details of a log group from the application server's Repository.	>dlgl <log_group_name>
open_log_group, openlg	Opens a log group for editing from the application server's Repository.	>openlg <log_group_name>
remove_log_group, rmlg	Deletes a log group from the application server's Repository.	>rmlg <log_group_name>
rename_log_group, renlg	Renames a log group in the application server's Repository.	>renlg <old_group_name> <new_group_name> Note: Maximum length for name is 32 characters.
xml	Note: This command is not used with the Log Groups utility.	
read_file, read	Directs command input to another input file. For example, you can direct input to run a batch file that contains a series of commands executable by this utility.	>read <filename>
silent	Note: This command is not used with the Log Groups utility.	
quit, q	Ends the program.	>quit

Examples

Creating a Log Group

The following table shows a sample log group creation, with a description of each entry.

Command Option:	Description:

Command Option:	Description:
>lgtool logger_a	Invokes the Log Group utility and optionally specifies logger_a as the application server. If an application server is not specified, the local machine is used.
>clg my_log_group	Initiates a request for a new group and specifies the group name.
>lsys	Displays a list of systems, instances, nodes, and processes whose logs can be added to the group.
>al /product ICM /system 192.168.2.199 /icm_customer customer_1 /icm_node LoggerA /icm_process csfs	Adds an ICM log to the group request.
>al /product CCS /system CollServer001	Adds a second (CCS) log to the group. Each log must be added on a separate line.
>lgdesc my_group_description	Optionally, creates a description of the group.
>closelg	Closes the log group and saves it to the application server's Repository.

See Also

For related information, see:

About Using Cisco Web Utilities from a Command Line

How to Use the Log Collector

How to Use the Log Collection Utility from a Command Line

How to Use the Log Collection Utility from a Command Line

Use the *Log Collection* utility to:

- Create a log collection
- Download collected logs
- View details of log collections
- Delete log collections

Note: The Log Collection utility can be run in interactive mode only; command-line mode is not available for this tool. For additional information on basic Web Tools command line use, including the distinction between command-line mode and interactive mode, see About Using Cisco Web Utilities from a Command Line.

To Run the Log Collection Utility from a Command Line

To run the Log Collection utility from a command line:

1. From a command prompt on a Support Tools node, navigate to `<support_tools_root>`.
2. Enter `lctool`. Optionally in the same line, to specify a different machine to serve as the application server, enter its DNS or IP address. If none is specified, the local machine serves as the application server. Press `<Enter>`.
3. Enter commands as described in the Command Line Options section below.

Using the Log Collection Utility from a Command Line

Command Line Options

Command:	Description:	Example:
<code>cmdhelp, chelp</code>	Displays a list of commands specific to this utility. Note: Using <code>Help</code> or <code>?</code> also displays this list, but includes several additional ICM commands (e.g., <code>echo</code> , <code>error_stop</code>) not used by this utility.	<code>>cmdhelp</code>
<code>/?</code>	Displays syntax for a specified command.	<code>><command> /?</code>

Command:	Description:	Example:
appserver	Optionally Specifies the system on which the utility should run. If not specified, the utility is run on the local system.	>lctool <app_servername>
system	Note: This command is not used with the Log Collection utility.	
localhost	Note: This command is not used with the Log Collection utility.	
create_collection_request, ccolreq	Opens a session for creating a log collection request.	>ccolreq <new_collection_name> Note: Maximum length for name is 32 characters
add_log_group, alg	Adds one log group from the application server's Repository into current log collection. Each log group must be added on a separate line.	>alg <log_group_name>
remove_log_group, rmlg	Removes 1 log group from current log collection.	>rmlg <log_group_name>
list_log_groups, lslg	Lists the log groups in the current collection request.	>lslg
add_request_description, lcdesc	Optionally, sets a description for current log collection.	>lcdesc <description>
set_icm_binary, bin /off /on	Optionally, sets the icmBinary flag ON or OFF. ON causes ICM logs to include binary data. If not specified, defaults to off.	>bin /on
set_log_range, range /start_date /start_time /end_date /end_time /all	Specifies the date/time range for logs to collect. /start_date, /start_time, /end_date and /end_time are all required. Date format is mm/dd/yyyy. Time format is hh:mm. Optionally, you can specify /all in place of specific dates/times.	>range /start_date mm/dd/yyyy /start_time hh:mm /end_date mm/dd/yyyy /end_time hh:mm
set_merge_logs, merge /off /on	Optionally, set merge flag ON or OFF. On causes logs to be merged. If not specified, defaults to off.	>merge /on
submit_collection_request, submit	Submits the current collection request to application server to start the log collection.	>submit
discard_collection_request, discard	Discards and closes the current collection request without exiting the Log Collection utility.	>discard
list_logs, lslc	List information, including status, of all log collections in the application server's Repository.	>lslc
view_collection, view	Displays detailed information of a specified log collection.	>view <collection_name>
remove_log, rmlc	Deletes log collection from the application server's Repository.	>rem <collection_name>

Command:	Description:	Example:
rename_log, ren	Renames a log collection in the application server's Repository.	>ren <old_collection_name> <new_collection_name> Note: Maximum length for name is 32 characters
download_log, download /filename /index	Downloads a specified log file from the application server's Repository to a local drive.	download
xml	Note: This command is not used with the Log Collection utility.	
read_file, read	Directs command input to another input file. For example, you can direct input to run a batch file that contains a series of commands executable by this utility.	>read <filename>
silent	Note: This command is not used with the Log Collection utility.	
quit, q	Ends the program.	>quit

Examples

Creating a Log Collection Request

The following table shows a sample log collection request, with a description of each entry.

Command Option:	Description:
>lctool	Invokes the Log Collection utility. Because no application server is specified, the local machine is used.
>ccolreq my_collection_name	Initiates a request for a new collection and specifies the collection name.
>alg log_group_1	Adds a log group from the application server's Repository to the request.
>alg log_group_2	Adds a second log group to the request. Each group must be added on a separate line.
>lcdesc my_collection_description	Optionally, creates a description of the request.
>range /start_date 09/01/2003 /start_time 00:00 /end_date 09/01/2003 /end_time 00:00	Specifies the date/time range for logs to collect. /start_date, /start_time, /end_date and /end_time are all required. Date format is mm/dd/yyyy. Time format is hh:mm. Optionally, you can specify /all in place of specific dates/times.
>bin on	Optionally, specifies that binary data should be included in the logs. If not specified, defaults to Off.

Command Option:	Description:
	Note: Applies to ICM products only. CCM, CMB, CEM, and CCS are not affected by this option.
>merge on	Optionally, specifies that logs should be merged. If not specified, defaults to Off.
>submit	Submits the request.

Downloading Collected Logs

The following table shows a sample log download request, with a description of each entry.

Note that you must download a log in order to view it. You cannot view collected logs directly from the Repository.

Logs download to the local machine, regardless of whether this is the machine currently serving as your application server.

Command Option:	Description:
>lctool	Invokes the Log Collection utility.
>ccolreq my_collection_name	Initiates a request for a new collection and specifies the collection name.
>view my_collection	<p>Displays all of the logs collected by a particular collection. You must use the view command prior to downloading a file.</p> <p>Returns output such as shown below. Note the unique index number (0, 1,2) prefacing each log in the collection:</p> <p>Name: "my_collection"</p> <p>Creation time: Thu Sep 2 11:28:17 2003</p> <p>Description:</p> <p>Merge: false</p> <p>Binary for ICM Log: false</p> <p>Selection type: all</p> <p>Including Log Group:</p> <p>[my_group]</p> <p>Collected item:</p> <p>0) ICM : 10.86.128.84 nam50, RouterA, dba</p> <p>1) ICM : 10.86.128.84 nam50, RouterA, et</p> <p>2) ICM : 10.86.128.84 nam50, RouterA, mds</p>

Command Option:	Description:
<pre>>download /index 0 /filename c:\my_collection.zip</pre>	<p>Specifies that the log whose index value is 0 should be downloaded from the Repository and saved to the local C drive in a file named my_collection.zip.</p> <p>Note that:</p> <ul style="list-style-type: none"> • To download all logs in the collection, omit the/index option. • You can name the downloaded file anything you like. Its file type, regardless of the extension specified, is always Zip.

See Also

For related information, see:

[About Using Cisco Web Utilities from a Command Line](#)

[How to Use the Log Collector](#)

[How to Use the Log Groups Utility from a Command Line](#)

Section IV. Using Cisco Common Utilities

About Cisco Common Utilities

Cisco common utilities are a set of Cisco-authored utilities designed for use with the ICM components supported by Support Tools 1.0(1).

Accessing Cisco Common Utilities

Most Cisco common utilities are command-line based, and can be run from either the Support Tools Dashboard or from an individual node. Command-line access to individual nodes can be local or remote (via methods like TelNet, pcAnywhere, etc.).

Several Cisco common utilities are GUI-based. While menu commands for these utilities appear in the Support Tools Dashboard, they cannot in fact be launched from there. Access to these tools is limited to the individual node (either locally or via remote GUI such as pcAnywhere). These GUI-based Cisco tools include:

- icmdba
- rtrtrace
- ss7nictrace

Cisco Common Utilities Locations

For the most part, Cisco Common Tools are installed on core ICM components only: AWs, PGs, Call Routes, and Loggers. Not every utility is installed on every node. For example, CICMan is only present on NAM Call Routers. See the table in the next section for more information.

If you attempt to launch a utility that is not present on the current system, the Support Tools Dashboard returns a response identical to the response you would get if you tried to spawn a non-present utility from a command prompt.

Cisco Common Utilities at a Glance

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	External GUI - No Dashboard or Command-Line Access:
cicman	Use on a NAM CallRouter to view and set NAM to CICM session information.	NAM Call Routers		
ctitest	Use for debugging or demonstration in a Cisco ICM CTI environment. Can be used in place of an Interactive CTI client application.	CTIOS PGs	X	
dbdiff	Third-party (Microsoft) tool to compare database tables. Invoked with the diffconfig.bat tool.	Nodes on which SQL Server is installed		
dumpcfg	Dump Config ICM audit utility. Use to analyze the ICM Config_Message_Log table. Allows you to determine WHO did WHAT, WHEN, from WHERE, using WHICH program.	Loggers (but can be run from any ICM component)		
icmdba	ICM Database Administration tool. Use to create, monitor, and edit ICM databases, including Logger, HDS, and AWDB databases. ICMDBA is also used to manage various SQL Server operating parameters.	AWs, Loggers (ICM components with DBs)		X
mptrace	Use to provide a playback from the Nortel Meridian Automatic Call Distributor (ACD) to troubleshoot potential issues.	MerPim PGs		
nicroi	NIC Remote Operator Interface. Use to configure and debug the (old-style DOS-based) AT&T NIC. Often invoked with the NIC.BAT batch program.	Routers for customers running an AT&T NIC	X	
nmstart	Node Manager Start. Use to start an ICM service on a CallRouter, Logger, AW, or PG.	AWs, PGs, Call Routers, Loggers	X	
nmstop	Node Manager Stop. Use to stop an ICM Service on a CallRouter, Logger, AW, or PG.	AWs, PGs, Call Routers, Loggers	X	
opctest	Use to interpret a Peripheral Gateways (PG) status, statistics, etc. It is also possible to enable specific debug tracing in the OPC process.	PGs	X	

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	External GUI - No Dashboard or Command-Line Access:
procmon	General-purpose debugging tool. Can be used in conjunction with various Network Interface Controller (NIC) and Peripheral Interface Manager (PIM) processes. Use to verify status, set debug trace bits, etc.	PGs, Call Routers	X	
rtrtrace	Use to set various debug tracing on a CallRouter. The additional tracing is output to the .EMS log files, and viewed with the Log Collector.	Call Routers		X
rttest	Use to interpret a Call Router's events and states live without interruption to the running router processes. Rttest has several subroutines that allow viewing status, statistics, etc. It is also possible to enable specific debug tracing in the call router.	Call Routers (but can be run from any ICM component)	X	
ss7nictrace	Use to view and set various debug trace bits on a new style NT SS7 NIC PG.	PGs		X
vrutrace	Use to output tracing information from a Voice Response Unit (VRU) device and to and from its PIM process. VRUTrace allows capture and playback of session data.	VRU PGs		



How to Use the CICMan Utility

Use the *CICMan* utility to view and set NAM to CICM session information for a NAM CallRouter. For use with NAM Call Routers only.

To Access CICMan from the Dashboard

To run CICMan from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > CICMan.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `cicman` command is already implied. Do not enter it in the Arguments field.
4. If desired, in the Commands field, enter (or paste from a batch file) a group of commands to run in batch mode. When you do this:
 - Separate individual commands by a new line.
 - Do not enter input flags.
 - Do not enter an input file name in the arguments field.
5. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
6. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
7. Click the Run button.

To Access CICMan from a Command Line on a Node

From a command-line on a NAM Call Router, you can run CICMan from the ICM root directory. For example:

```
c:\><icm_root>\cicman
```

Using CICMan

Command Line Options

Syntax: `cicman [/f InputFile] [/system SystemName] [/cust Customer] [/node ICMNode] [/pipe OutputPipe] [/debug] [/stop] [/help] [/?]`

Parameter Descriptions

`cicr_meters, cmeters`: Displays the CICR meters for both CIC sides.

`echo`: Controls echoing of command lines

`error_stop`: Controls setting of stop on error flag.

`glbl_meters, gmetrics`: Displays the global meters for both CIC sides.

`help, ?`: Display program description.

`link_ctrl, lctrl`: Used in upgrade to shut down all links 1-on a CIC A/B or 2-to all CICRs side A/B.

`link_stat, lstat`: Displays the link status for both CIC sides.

`quit, q`: Ends the program.

`read_file, read`: Directs command input to another input file

`table_values, tbl_vals`: Displays the CIC Table values including timeouts & thresholds.

See Also

For related information, see:

About Cisco Common Utilities

How to Use the CTITest Utility

Use the *CTITest* utility to debug in a Cisco ICM CTI environment. CTITest can be used as an interactive CTI client for demonstration or debugging purposes.

It connects to the CTI Server through a socket connection, and displays either all of the events that are occurring on the switch (a CTI bridge client application), or only the events which pertain to a single agent. If a CTITest has the same configuration that your application does, the expected events and switch behavior is seen in the CTITest program.

For use with CTIOS PGs only.

To Access CTITest from the Dashboard

To run CTITest from the Support Tools Dashboard:

1. Use the System Select screen to select the system (namely, PG) you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > CTITest.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `ctitest` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

To Access CTITest from a Command Line on a Node

From a command-line on a CTIOS PG, you can access CTITest from any location on the local physical drive. For example:

```
c:\>ctitest
```

```
d:\temp>ctitest
```


Using CTITest

Starting CTITest requires:

- The CTI Server (PG) IP address
- The socket number
- The agent ID and login information

Note: In addition to `ctitest.exe`, `trace.dll (icrmmsgs.dll)` is also located in the `<icm_root>\bin` directory. Because `icrmmsgs.dll` is an active services file, it is recommended that these two files be copied into separate directory.

Configuring CTITest

Configure the CTITest as described in following steps. In the example below, `abc` is the separate directory as outlined above.

1. Set up a profile which consists of settings used as input to CTITest. To initiate a new profile, start the CTITest with the following command:

```
c:\abc\>ctitest /p <profile name>
```

2. Configure the CTI Server and socket where the CTITest connects to:

- a. Configuring Side A CTI Server: Run the following command to configure the CTI Server Side A for the CTITest.

- b. `c:\abc\>ctitest: config /hostA <ctiserver_hostnameA> /portA <ctiserver_portA>`

- c. Configuring Side B CTI Server: Run the following command to configure the CTI Server Side B for the CTITest if you have a duplex configuration.

- d. `c:\abc\>ctitest: config /hostb <ctiserver_hostnameB> /portB <ctiserver_portB>`

Note: In the above example, `<ctiserver_hostnameA>` `<ctiserver_hostnameB>` represents the IP address or host name of the CTI Server or PG side A and B, and `<ctiserver_portA>` `<ctiserver_portB>` represents the port number of side A and B.

Note: If it is a simplex environment, use "" to specify a null value.

3. Configure the CTI protocol version to 6:

```
c:\abc\>ctitest: config /version 6
```

4. Configure the service mask to 7:

```
c:\abc\>ctitest: config /service 7
```

5. It is suggested that you turn off heartbeats while debugging the CTI issues. To configure heartbeat to -1, use the command:

```
c:\abc\>ctitest: config /hb -1
```

6. Configure the agent to determine the desired Client Events for Client Mode. These vary depending on your CTI Server environment:

```
c:\abc\>ctitest: agent /periph <peripheral_id> /id <agentid>
/password <password> /ext <extension> /inst <instrument>
```

Note: If there is no agent password required to login, use "" to specify a null value. Run the agent /? command for help.

Opening a Session

The open command performs a socket connection to the CTI Server, and issues the appropriate commands to the CTI Server to establish message communication. By default, the CTITest repeatedly attempts to open a session until it either opens a session, or the close or quit command is given. It automatically attempts to connect to the other side following a failure of the CTI Server. You can open a session after completing a configuration successfully as described below.

```
c:\abc\>ctitest: open
```

If your CTITest configuration is correct, a message appears similar to the following:

```
session #2 opened after 0 ms CTime 13:52:32 PGStatus NORMAL Peripheral:
Online
```

If errors are received while opening a session, verify your CTITest configuration as outlined above.

Logging In

You can login into the Automatic Call Distributor (ACD) that the PG is communicating with, if the session is successfully opened. Below is the syntax for issuing the login command:

```
Syntax: CTITest: login /? usage: agent_login ID PASSWORD [/periph N]
[/instrument N] [/reason N] [/position N] [/supervisor N] [/auctoIN]
[/manualIN] [/skill N[.pri] [/skill N[.pri]] ]
```

Based on the type and configuration of the ACD, you need to provide these parameters for a successful login. Once connected, you can change agent states, answer call, transfer call, make call, conference, etc. using CTITest.

Command Line Options

Below is list of frequently used commands in CTITest:

Command:	Parameters:
agent_login (login)	ID PASSWORD [/periph n] [/instrument N] [/reason N] [/position N] [/supervisor N] [/autoIN] [/manualIN] [/skill N[.pri]] [/skill N[.pri]] ...]
agent_logout (logout)	[/agentID ID] [/password] [/periph n] [/instrument N] [/reason N] [/skill N[.pri]] [/skill N[.pri]] ...]
agent_not_ready (not_ready)	[/agentID ID] [/periph n] [/instrument N] [/reason N] [/skill N[.pri]] [/skill N[.pri]] ...]
agent_ready (ready)	[/agentID ID] [/periph n] [/instrument N] [/reason N] [/autoIN] [/manualIN] [/skill N[.pri]] [/skill N[.pri]] ...]
agent_work_not_ready (wrapup)	[/agentID ID] [/periph N] [/instrument N] [/reason N] [/skill N[.pri]] [/skill N[.pri]] ...]
agent_work_ready (callwork)	[/agentID ID] [/periph N] [/instrument N] [/reason N] [/skill N[.pri]] [/skill N[.pri]] ...]
answer_call (answer)	[/periph N] [/callid CALLID.DEVID] [/stack N] [/instrument N]
autoanswer	[/on] [/off]
clear_call (clear)	[/periph N] [/callid N.devid] [/stack N]
conference_call (conf)	[/periph N] [/active CALLID.DEVID] [/held CALLID.DEVID] [/stack ACTIVE HELD] [/blind DN] [/placement TYPE] [/manner TYPE] [/alertRings N] [/callopt TYPE] [/TG N] [/SG N] [/authorize code] [/account code]
consult_call (cc)	DN [/periph N] [/instrument N] [/callid N.devid] [/stack N] [/placement TYPE] [/manner TYPE] [/transfer] [/conference] [/alertRings N] [/callopt TYPE] [/priority] [/postroute] [/TG N] [/SG N] [/authorize code] [/account code]
get_agent_state (state)	[/periph n] [/instrument N]
get_agent_stats(stats)	[/periph N] [/instrument N] [/ext string] [/id string]
get_skill_stats (sk_stats)	[/periph N] [/skill N] [/skillID N]
hold_call (hold)	[/periph N] [/callid N.devid] [/stack N]
make_call (mc)	DN [/periph N] [/instrument N] [/placement TYPE] [/manner TYPE] [/alertRings N] [/callopt TYPE] [/priority] [/postroute] [/TG N] [/SG N] [/authorize code] [/account code] [keyword=VALUE ...]
release	[/periph N] [/callid N.devid] [/stack N]
retrieve_call (retrieve)	[/periph N] [/callid N.devid] [/stack N]
transfer_call (transfer)	[/periph N] [/active CALLID.DEVID] [/held CALLID.DEVID] [/stack ACTIVE HELD] [/blind DN] [/placementTYPE] [/manner TYPE] [/alertRings N] [/callopt TYPE] [/TG N] [/SG N] [/authorize code] [/account code]

See Also

For related information, see:

About Cisco Common Utilities
The Cisco ICM CTI Programmer's Guide
Cisco TAC ICM Technical Notes on Cisco.com

How to Use the DBDiff Utility

Use the *DBDiff* utility to perform comparisons of database tables from two different databases. For example, you might want to check that the ICR_Locks table contains the same data on both sides of a Central Controller.

For use with nodes on which SQL Server is installed only.

To Access DBDiff from the Dashboard

To run DBDiff from the Support Tools Dashboard:

1. Use the System Select screen to select the system (namely, PG) you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > DBDiff.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `dbdiff` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

To Access DBDiff from a Command Line on a Node

From a command-line on an ICM node on which SQL Server is installed, you can access DBDiff from any location on the drive on which ICM is installed. For example:

```
c:\>dbdiff  
c:\temp>dbdiff
```

Using DBDiff

Syntax: `dbdiff <database1.table@host1> <database2.table@host2>`

For example: `dbdiff cust1_sideA.ICR_Locks@geoxylgra
cust1_sideB.ICR_Locks@geoxylgrb`

You can also use the batch script `diffconfig.bat` to invoke DBDiff for various tables to automatically compare two ICM databases.

Syntax: `diffconfig <database1> <host1> <database2> <host2>
<database2.table@host2>`

For example: `diffconfig cust1_sideA geoxylgra cust1_sideB geoxylgrb`

Command Line Options

Syntax: `DBDiff {database a}. {table a}[@server a] {database b}. {table
b}[@server b] [/out:{file}] [/key:{pkey1,pkey2,...}] [/where:"{where
clause}"]`

See Also

For related information, see:

About Cisco Common Utilities

The Cisco ICM Administrator Guide

How to Use the DumpCfg Utility

Use the *DumpCfg* utility to analyze records in the ICM Config_Message_Log table to determine what actions have been performed on an ICM system, when, by whom, and using what applications.

For use with ICM Loggers only, but can be run from any ICM component.

To Access DumpCfg from the Dashboard

To run DumpCfg from the Support Tools Dashboard:

1. Use the System Select screen to select the system (namely, PG) you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > DumpCfg.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `dumpcfg` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

To Access DumpCfg from a Command Line on a Node

From a command-line on an ICM Logger, you can access DumpCfg from any location on the local physical drive. For example:

```
c:\>dumpcfg
```

```
d:\temp>dumpcfg
```

Using DumpCfg

Syntax: `DumpCfg <database>[@server] <low recovery key> [high recovery key]`

See Also

For related information, see:

[About Cisco Common Utilities](#)

How to Use the ICMDBA Utility

Use ICMDBA, the ICM Database Administration tool, to create, monitor, and edit ICM databases, including Logger, HDS, and AWDB databases. You can also use ICMDBA to manage various SQL Server operating parameters. You can use ICMDBA to:

- Estimate size and bandwidth requirements for databases.
- Create, edit and delete central databases, local databases, and historical database for installed ICM customers.
- Resize database devices.
- Recreate a database.
- Import/export data to/from databases.
- View database properties.

In addition, you can use ICMDBA to start or stop a server, and to do some limited SQL server configuration.

ICMDBA is for use with any ICM component that uses a database (namely, AWs, Loggers).

Accessing ICMDBA

ICMDBA is an external GUI application present on all ICM nodes as part of that standard ICM installation. It cannot be run from within the Support Tools dashboard.

To access ICMDBA on an ICM component which uses a database (namely, AWs, Loggers), from a command line, enter `icmdba`.

See Also

For related information, see:

About Cisco Common Utilities

Cisco ICM Administrator Guide

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the MPTrace Utility

Use the *MPTrace* utility to get a playback from the Meridian ACD to troubleshoot potential issues. MPTrace provides information on what is configured on the ACD and what needs to be changed. It looks at Controlled Directory Numbers (CDNs) and agent Position IDs/Instrument Identification Numbers (IDNs).

For use with MerPim PGs only.

To Access MPTrace from the Dashboard

To run MPTrace from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > MPTrace.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `mptrace` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

To Access MPTrace from a Command Line on a Node

From a command-line on an ICM MerPim PG, access MPTrace from the `merpimcap` directory located at: `<icm_root>\<customer instance>\<pg#>`

For example:

```
c:\icm\my_customer_instance\pg1a\merpimcap> mptrace
```

Using MPTrace

The `mptrace` command places two text files in the `mptrace` directory:

- The `pim1_summary` file is a playback of the ACD configuration and includes information on unconfigured Position IDs and CDNs, as well as Meridian Link misconfigurations.
- The `pim1_trace` file is the complete dialog of what the ACD is seeing during this capture.

Note: The `pim1_summary` and `pim1_trace` files are overwritten every time MPTrace is used.

When adding new configurations to the ACD, old `.cap` files from the `merpimcap` directory should be deleted so that you can see fresh results after new `.cap` files are written. The `.cap` files are the raw binary files that are written from the ACD to the `merpimcap` directory. When an MPTrace is run, these binary `.cap` files are converted into two text files. If there are old files in the directory, then the playback of the configuration includes old data that may have already been addressed and fixed.

Command Line Options

Syntax: `MPTrace [ProcessName(s)] [/if InputFile] [/tf TraceOutputFile] [/sf SummaryOutputFile] [/bd BeginDate(mm/dd/yyyy)] [/bt BeginTime(hh:mm:ss)] [/ed EndDate(mm/dd/yyyy)] [/et EndTime(hh:mm:ss)] [/hr HoursBack] [/all] [/last] [/prev] [/bin] [/ms] [/debug] [/regbase RegBase] [/icrroot icmRoot] [/pbreal] [/pb_sb PlaybackSecondsToBackup] [/pb_sync hh] [/pb_sync_today] [/pb_fsph PlaybackFileStartPositionHint] [/nobuf] [/notracemlink] [/notracemei] [/tracemlkhex] [/tracemeihex] [/notracerd] [/notracetp] [/help] [/?]`

See Also

For related information, see:

About Cisco Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the NICROI Utility

Use the *NICROI* (NIC Remote Operator Interface) utility to configure and debug the (old-style DOS-based) AT&T NIC. NICROI allows you to see route request data from the inter-exchange carrier (IXC), and label response information from the Cisco Intelligent Contact Management (ICM) Call Routers.

For use with ICM Loggers for customers running an AT&T NIC only.

To Access NICROI from the Dashboard

To run NICROI from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > NICROI.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `nicroi` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

To Access NICROI from a Command Line on a Node

From a command-line on an ICM Logger for customers running an AT&T NIC, you can access NICROI by entering:

Method 1

```
nic <hostname> 5300 support <customer_instance>xyzzyf
```

For example:

```
c:\>nic my_host_name 5300 support my_customer_instancexyzzy
```

Note: In the example above, `support` and `<customer_instance>xyzyyf` represent respectively the out-of-box admin username and password for this utility. Make appropriate substitutions if these values have been modified.

Method 2

```
nic <customer_instance> <nic_extension>
```

For example:

```
c:\>nic my_customer_instance 1a
```

Using NICROI

Using Tracing in NICROI

There is only one level of tracing available within NICROI, it allows you to view Route Requests from the carrier. These route requests include the caller's Calling Line ID, any Caller Entered Digits, and the Dialed Digits (same as Dialed Number).

To turn on tracing:

Type `nic sfk 4` in the NICROI CONSOLE window. (There is not a prompt in this window, just an empty command line where you may enter commands.) This is the only trace element available within NICROI. You should now be able to click on the activity window behind your NICROI session and see the results of this trace setting scrolling.

To turn off tracing:

In order to turn tracing off, repeat the same entry `nic sfk 4`.

This added information is then inserted into a log file, either the `niclog.xxx` or the `roilog.txt` log file that you open/create. (explained below).

Log Files

There are two different places that you can capture this data, within the `niclog.xxx` or within a log file that you must open in order to have data written to it called `roilog.txt`.

Niclog.xxx File

The `niclog.xxx` is a file that the NIC writes data to automatically and labels each log file with a date/time stamp, along the same functionality as an EMS log file used for other ICM processes. Use the following steps to view a list of `niclog.xxx` files:

1. Type `mgmt help` within the NICROI window to see the options available.
2. Type `mgmt file list` to view all the `niclog.xxx` files available.
Note: The time shown is the time that the log is closed, meaning the time that information stopped being written to that particular log and another log is created. You can see in this list also the size of the file.
3. If you do not see the date and time you are looking for, you can view more log files by typing `mgmt file list more` until you see the log file that you would like to capture. This log file format is useful if you have several specific time periods that you want to capture.

Roilog.txt File

The `roilog.txt` file is a file that is invoked by opening and closing it, this file is NOT automatically generated as the `niclog.xxx`. This log file is useful if you have one general, extended period of time that you want to capture. Use the following steps to create a `roilog.txt` file:

To "Open" a `roilog.txt` file:

From within NICROI issue the `log open [<filename>]` command. This command causes all status displayed in the NICROI logging window to be stored in a log file on the NICROI host system. The name of the log file can be specified. If omitted, the default name is `roilog.txt` which is placed in the current directory of the process from which NICROI starts.

Note: If you use this type of log file, you must close the log to stop the writing of information (once you have captured the time frame in question) before you copy the log file out to analyze it.

To "Close" an `roilog.txt` file:

From within NICROI type the `.log` command to close the NICROI log file. No further file logging is performed.

Copying Log Files

You can copy these log files to your local drive. There are basically two ways you can gather this data, based on which file format you choose to use above.

Copying Files Using `Roilog.txt`

If you have opened and created a log file using the method for `roilog.txt`, upon closing the log file, you have a copy of the log on your local drive. The file will be located in the directory from which you began your NICROI session. Unless you have specified a download directory using the `.xdir c:\<path name>` command, then the file is created and written onto your local drive at the location from which you started your NICROI session.

Copying Files Using `Niclog.xxx`

If you choose to use the `niclog.xxx` files you need to download or transfer the files to your local drive using the `nmg roi transfer` command as outlined below. Remember, if you do not specify a transfer directory with the `.xdir c:\<path Name>` command then the file is transferred to the directory from which you started your NICROI session. The following two sections are examples of these download commands.

Setting the Download Directory

The `.xdir d:\support` command allows you to set your download directory so all `roilog.txt` files created are automatically transferred to that directory as well as any `niclog.xxx` files downloaded using the `nmg roi transfer` command (syntax below). If you do not use this command you may still gather these log files. They are then automatically downloaded into the directory from which you started your NICROI session.

Transferring Files

The `nmg roi transfer <file name>` command allows you to transfer `niclog.xxx` files from NICROI to your local physical drive. Once you execute this command you can see the following lines from within your NICROI session as confirmation that the file transferred correctly.

```
nmg roi transfer <file name>
->BEGIN FILE TRANSFER of <file name>
->TRANSFER: d:\support\<file name> <file size> <date / time / year>
->FILE TRANSFER COMPLETE, seqnum = 3
```

Examples

Examples of `roilog` with and without SFK 4 tracing are as follows:

Without SFK 4 Tracing:

```
1056901832 14:53:11 CCComm: (A)DMPRecvMsg: len=56, class=131, type=9,
msg_buf=9CA9:0672
1056901833 14:53:11 ICRGate: RecvCRSMail: class = 131, len = 56, buf =
9CA9:0672
1056901973 14:53:11 CCComm: DMPReturnMsg: len=72, class=131, type=8,
msg_buf=8CA8:3B2E
1056902004 14:53:11 CCComm: (A)DMPRecvMsg: len=56, class=131, type=9,
msg_buf=9CA9:0672
1056902005 14:53:11 ICRGate: RecvCRSMail: class = 131, len = 56, buf =
9CA9:0672
1056902062 14:53:11 EMT: (0) Hearbeat sent. Countdown=4
1056902093 14:53:11 CCComm: DMPReturnMsg: len=72, class=131, type=8,
msg_buf=8CA8:3DCE
1056902124 14:53:11 CCComm: (A)DMPRecvMsg: len=56, class=131, type=9,
msg_buf=9CA9:0672
```

```
1056902124 14:53:11 CCComm: DMPSendAck: hi-seq=2244083712, lo-  
seq=2151022592  
1056902125 14:53:11 ICRGate: RecvCRSMail: class = 131, len = 56, buf =  
9CA9:0672  
1056902163 14:53:11 EMT: (0) Hearbeat received.
```

With SFK 4 Tracing:

```
1057065593 14:54:45 MTP3: DIAG SLTC[0]: (<- HMDT (SLTM)) -> HMRT (SLTA)  
1057065681 14:54:45 ICRGate: (5071) XId=0x10010(396), Q=1,  
DD=8002220300  
1057065681 14:54:45 ICRGate: (5071) ANI=5089557093, CED=  
1057065682 14:54:45 CCComm: DMPReturnMsg: len=72, class=131, type=8,  
msg_buf=8CA8:430E  
1057065697 14:54:45 CCComm: (A)DMPRecvMsg: len=56, class=131, type=9,  
msg_buf=9CA9:068E  
1057065697 14:54:45 CCComm: DMPSendAck: hi-seq=2991849473, lo-  
seq=2952986624  
1057065698 14:54:45 ICRGate: RecvCRSMail: class = 131, len = 56, buf =  
9CA9:068E  
1057065698 14:54:45 ICRGate: (5071) (396) Label=7846402130, Type=0,  
CDPD=  
1057065738 14:54:45 EMT: (0) Hearbeat received.  
1057065819 14:54:45 ICRGate: (5071) XId=0x10001(397), Q=1,  
DD=8002220300  
1057065819 14:54:45 ICRGate: (5071) ANI=2127410484, CED=3  
1057065820 14:54:45 CCComm: DMPReturnMsg: len=72, class=131, type=8,  
msg_buf=8CA8:45AE  
1057065831 14:54:45 CCComm: (A)DMPRecvMsg: len=56, class=131, type=9,  
msg_buf=9CA9:06AA  
1057065832 14:54:45 ICRGate: RecvCRSMail: class = 131, len = 56, buf =  
9CA9:06AA  
1057065832 14:54:45 ICRGate: (5071) (397) Label=7846402076, Type=0,  
CDPD=  
1057065966 14:54:45 EMT: (0) Hearbeat sent. Countdown=4  
1057066098 14:54:45 EMT: (0) Hearbeat received.
```

See Also

For related information, see:

About Cisco Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the NMStart Utility

Use the *NMStart* (Node Manager Start) utility to start an ICM service on an ICM Call Router, Logger, AW, or PG.

To Access NMStart from the Dashboard

To run NMStart from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > NMStart.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `nmstart` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

To Access NMStart from a Command Line on a Node

From a command-line on an ICM Call Router, Logger, AW, or PG, you can access NMStart from any location on the local physical drive. For example:

```
c:\>nmstart
```

```
d:\temp>nmstart
```

Using NMStart

Syntax: `nmstart <customer_instance> <node>`

For example: `c:\> nmstart cisco pg3a`

See Also

For related information, see:

[About Cisco Common Utilities](#)

[Cisco TAC ICM Technical Notes on Cisco.com](#)

How to Use the NMStop Utility

Use the NMStop utility to stop an ICM service on an ICM Call Router, Logger, AW, or PG.

Note: The Support Tools Unknown Process and Services Explorer is an alternate, more robust utility with which to stop ICM services.

To Access NMStop from the Dashboard

To run NMStop from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > NMStop.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `nmstop` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

To Access NMStop from a Command Line on a Node

From a command-line on an ICM Call Router, Logger, AW, or PG, you can access NMStop from any location on the local physical drive. For example:

```
c:\>nmstop
```

```
d:\temp>nmstop
```

Using NMStop

Syntax: `nmstop <customer_instance> <node>`

For example: `c:\> nmstop cisco pg3a`

See Also

For related information, see:

About Cisco Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the OPCTest Utility

Use the *OPCTest* utility to interpret a Peripheral Gateway's (PG) status, statistics, etc. It is also possible to enable specific debug tracing in the OPC process.

For use with ICM PGs only.

To Access OPCTest from the Dashboard

To run OPCTest from the Support Tools Dashboard: To run OPCTest:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > OPCTest.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `opctest` command is already implied. Do not enter it in the Arguments field.
4. If desired, in the Commands field, enter (or paste from a batch file) a group of commands to run in batch mode. When you do this:
 - Separate individual commands by a new line.
 - Do not enter input flags.
 - Do not enter an input file name in the arguments field.
5. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
6. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
7. Click the Run button.

To Access OPCTest from a Command Line on a Node

From a command-line on an ICM PG, you can access OPCTest from any location on the local physical drive by entering: `opctest /<customer_instance> /<node>`

For example:

```
c:\>opctest /my_customer_instance /pg1a
d:\temp>opctest /my_customer_instance /pg1a
```

Using OPCTest

To list available commands, type `help` or `?` from the OPCTest command prompt.

Some of the OPCTest commands, such as `List_Agents` and `List_Trunk_Group`, require one or more additional command line switches. Type `command name /?` to obtain the proper syntax.

Command Line Options

Syntax: `opctest [/f InputFile] [/system SystemName] [/cust Customer] [/node ICMNode] [/pipe OutputPipe] [/debug] [/stop] [/help] [/?]`

Parameter Descriptions

`agent_trace`: Controls agent tracing in the router.

`call_control`, `call`: Controls various kinds access to OPC calls.

`checksum`, `check`: Requests router to do a checksum.

`debug_control`, `debug`: Controls various kinds of debugging output from opc.

`dump`: Requests opc to do a internal state dump to a file.

`dump_agent`, `da`: Dump contents of Agent within OPC.

`dump_call`, `dc`: Dump OPCs call states.

`dump_hash`, `dh`: Display OPCs internal hash statistics.

`dump_struct`, `ds`: Requests OPC to dump a structure.

`dump_tpservices`, `dtps`: Dump Third Party Service Data object.

`echo`: Controls echoing of command lines

`error_stop`: Controls setting of stop on error flag.

`exit_opc`, `exitopc`: Sends an message telling the OPC to exit.

`expression`, `expr`: Gives the router an expression to evaluate.

`help`, `?`: Displays this help.

`list_agents, la`: Display OPCs AgentStates for specified Group.

`list_calls, lc`: Display OPCs call states.

`list_routing_client, lrc`: Display OPCs Routing Client stats for peripheral.

`mem_leaks`: Dump memory leakage based on checkpoints from prior calls.

`network_trunk_group_trace, ntg_trace`: Controls NetworkTrunkGroup tracing in the OPC.

`peripheral_trace`: Controls Peripheral tracing in the OPC.

`quit, q`: Ends the program.

`read_file, read`: Directs command input to another input file

`route_call`: Sends a route call request to the router.

`route_trace`: Controls Route tracing in the OPC.

`service_trace`: Controls Service tracing in the OPC.

`skill_group_trace`: Controls SkillGroup tracing in the OPC.

`status`: Sends an message telling OPC to display its status.

`stop_log`: Requests logmsg to stop logging messages.

`trunk_group_trace`: Controls TrunkGroup tracing in the OPC.

`tuning_param, tune`: Controls tuning in the router.

Example

The following example shows detailed output for the `status` command:

```

C:\>opctest /cust/node pgl a
OPCTEST Release 2.5 (service pack 2), Build 03105
opctest: status
OPC Version: Release 2.5 (service pack 2), Build 03116
Release Date: 09/28/98 07:01:57

Current Time: 03/17 17:47:07
Local Time: 03/17 12:47:07 (5.0 hr)
OPC Up: 12/08 18:59:52 (98.9 day)

OPC Sync: 03/13 22:18:33 (3.8 day) (A->B)

Process      LastStateChange      LastHeartBeat
A opc       H--                  03/17 17:46:52 (16 sec)
A pgag     OK M- 12/08 18:59:56 (98.9 day) --
A piml    OK M- 03/12 19:35:58 (4.9 day) --
A ctisvr   ---                  --
B pgag     OK M- 12/08 18:59:53 (98.9 day) --
B piml    OK M- 03/13 22:18:42 (3.8 day) --
B ctisvr   ---                  --
B opc     H--                  03/17 17:46:52 (16 sec)

PGAgent  LastStateChangeTime  ConnectATime      Status      ConnectETime      Status
SideA  PIA 03/13 22:18:32 (3.8 day) 03/13 22:18:32 (3.8 day) CONNECTED 03/13 22:18:32 (3.8 day) CONNECTED
SideB  P-- 03/13 22:18:32 (3.8 day) 03/13 22:18:32 (3.8 day) CONNECTED 03/13 22:18:32 (3.8 day) CONNECTED

PeripheralID Side      State      LastStateChange      LastHeardFrom
1          A      PIM_ACTIVE PR 03/13 22:18:32 (3.8 day) 03/17 17:47:07 (1 sec)

CTIServerNo Side      State      LastStateChange      LastHeardFrom
1          ?      CTI_NULL   12/08 19:00:02 (98.9 day) --

```

Indicates which side of the PG is supplying the Call Router with status of the PG.

Processes on the PGs & their states.

PGAG – manages session layer communications between the PG & the Central Controller
SEE NOTE BELOW

Peripheral Interface Manager/PIM

CTI Server

Debug Information

You can enable specific debugging within OPCTest by issuing the debug command. Turning up specific tracing is much more effective than going into the registry, or turning up the EMSTraceMask for the entire OPC process. Enabling debug control turns up tracing on the part of OPC for that you need additional tracing. The tracing result is displayed in the OPC EMS log files. Use the Log Collector utility to view the output of the EMS logs.

For example:

```

opctest: debug /? Usage: debug_control [/realtime] [/agent] [/halfhour]
[/rcmeter] [/routing] [/skillgroup] [/closedcalls] [/cstaacr] [/cstacer]
[/pimmsg] [/ctimsg] [/rcmsg] [/dmpmsg] [/icmsg] [/opcmsg] [/mdsmg]
[/pdmsg] [/inrcmsg] [/passthru] [/tpmsg] [/physctrlr] [/periph] [/all]
[/help] [/?]

```

If you need to troubleshoot a translation route problem, try the debug /routing command.

Note: the /noall switch. Leaving tracing turned up can cause performance problems.

Exiting and Quitting OPCTest

Use the `quit` command to exit OPCTest.

Caution: Use the `exit_opc` command with caution. This command instructs the OPC process (on both sides of the PG if duplexed) to exit. Node Manager forces the process to restart, which then forces it to reload the configuration for the Call Router. All internal peripheral and agent states are flushed, then OPC and PIM re-learn the PG and its configuration.

See Also

For related information, see:

About Cisco Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the Log Collector Utility

How to Use the Procmon Utility

Use the *Procmon* utility for general-purpose ICM command-line debugging. It can be used in conjunction with various NIC and PIM. processes. Procmon allows you to perform actions like verifying status, setting debug trace bits, and so on.

For use with ICM PGs and Call Routers only.

To Access Procmon from the Dashboard

To run Procmon from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Procmon.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `opctest` command is already implied. Do not enter it in the Arguments field.
4. If desired, in the Commands field, enter (or paste from a batch file) a group of commands to run in batch mode. When you do this:
 - Separate individual commands by a new line.
 - Do not enter input flags.
 - Do not enter an input file name in the arguments field.
5. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
6. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
7. Click the Run button.

To Access Procmon from a Command Line on a Node

From a command-line on an ICM PG or Call Router, you can access Procmon from any location on the local physical drive. For example:

```
c:\>procmon
```

```
d:\temp>procmon
```

Using Procmon

Procmon supports both local and remote commands. Local commands are defined within Procmon itself, while remote commands are programmed into the monitored process. Below are lists of basic Procmon commands and process-specific commands for use with such processes as PIM, CTISVR and OPC.

Syntax: PROCMon CustomerName NodeName ProcessName [SystemName] [/f InputFile] [/wait] [/stop] [/help] [/?]

Basic Commands

Type `help` to display the list of basic commands described in the following table.

Command:	Definition:
echo	Controls echoing of command lines
emsmon	Command to control remote EMS monitor process (start, stop, pause, resume)
error_stop	Controls setting of stop on error flag
help, ?	Displays this help
monitor_help, mhelp	Displays Monitor Server help
monitor_sleep, msleep	Sleep for specified seconds or milliseconds
quit, q	Ends the program
read_file, read	Directs command input to another input file

Process-Specific and Troubleshooting Commands

Each Peripheral Type contains a different set of commands. For a list of commands associated with each peripheral, use `mhelp`. Each command has its own syntax. to determine the syntax, type the command followed by `/?`.

The table below lists commands used commonly during troubleshooting:

Command:	Definition:
pim_list_agents, la	List agents currently configured by PIM
pim_list_services, ls	List Services currently configured by PIM

pim_list_skill_groups, lsg	List SkillGroups currently configured by PIM
acd_debug, debug	Turn on/off debug trace
pim_list_trace, ltrace	List current PIM trace bit settings
pim_trace, trace	Set or reset PIM trace bits
pim_dump_periph, acdperiph	Dump contents of peripheral object
Quit	Ends the Procmon Program

Samples

This section examines some sample Procmon output.

```
>>>>la
```

```

SkillTarget ID      Periph#      C      Ext#      Inst#
ActGroups
          5000      6000 Y Yes      3000(3000) <1/ 1> [BO]
[265436], <2/ 2> [BO] [866278]
          5001      6001 Y Yes      3001(3001) <1/ 1> [AV]
[59704], <2/ 2> [AV] [59704]
          5002      6002 Y No        -1(-1)
          5003      6003 Y No        -1(-1)
          5028      6030 Y No        -1(-1)

```

```
>>>>ls
```

```

SkillTarget ID      Periph#      C      SerMem      Pri      SerTH      SLType
PSLType      Ext#
          5017      6500      Y          1          2          30          1
4
          5018      6501      Y          2          1          30          1
4
          5019      6502      Y          3          1          30          1
4

```

In the output above, SLType indicates the default value for the ServiceLevelType field for each service associated with the peripheral. This indicates how ICM calculates the service level. You can override this default for individual services.

PSLTYPE indicates the default value for the PeripheralServiceLevelType for each service associated with the peripheral. You can override this default for individual services.

```
>>>>lsg
```

```

      Periph#      Pri      C      SkillTarget ID      Ext#
          1          0      Y          5007          6900
          1          1      Y          5008          6900
          1          2      Y          5009          6900
          2          0      Y          5010          6901
          2          1      Y          5011          6901
          2          2      Y          5012          6901
          3          0      Y          5013          6902
          3          1      Y          5014          6902

```

3	2	Y	5015	6902
3	3	Y	5016	6902

```
>>>>debug /?
Usage: acd_debug [/noagent] [/agent] [/agent+] [/agent++] [/nobri]
[/bri] [/bri+]
        [/nocall] [/call] [/call+] [/nocms] [/cms] [/cms+]
[/csc]
        [/csc+] [/nocsc] [/noconfig] [/config] [/nocv] [/cv]
[/noerror]
        [/error][/nohb] [/hb] [/noopc] [/opc] [/nopost]
[/post] [/nosim]
        [/sim] [/notg] [/tg] [/notimer] [/timer] [/notp] [/tp]
[/tp+]
        [/trace] [/novq] [/vq] [/warning] [/nowarning] [/all]
[/noall]
        [/set UserSetBit] [/help] [/?]
```

```
>>>>debug /call+ /post /agent
Trace: AGENT CALL+ POST
UserTraceLevel=0xE848200003FFFFFF800E000000000000000000000000040
Time stamp: 09/13/02
```

Note: The debug feature only remains active while the process remains active. Once the process exits, the debug utility is no longer functioning. You can make the trace permanent by adding the hexadecimal number found in the UserTraceLevel line to the EMS trace in the registry.

```
>>>>acdperiph
BuildNum: 08799 (Rel 4.6.2) Time: 06/11/02 16:27:40
SwitchTime=08/26/02 13:56:22, DefRoute=CTIVarMap-NNNNNNNNNNN (y=PIM
access)CTIString=
CVBridge=[G3MsgRecvCnt=169239 (0x29517) Min/AllBrisUp=1/1
NumMonitored=1
PhysBris=0x1 RtBris=0x0 BadBris=0x0]
Bri[0] State=ACTIVE GoIdle=0
[NtwrkCngstn[Forced=F Switch=F]
Window=10000 MsgDlyTime=500
BriCfgParams(Exp.) = [*CvHost[0]=taclablg3 CvHost[1]= ]
Msgs [Sent=157095 (0x265a7) Recv=169239 (0x29517) ] [SA0id=314182
LastSA0IdRecvd=31482
Msgs [SendQ=0x0 SentQ=0x0 RecvQ=0x0 ]
Msgs [PriSendQ=0x0 RecvQ=0x0 ]
[ActiveAssoc[Avail=2033 Locked=11] OutstandingSent=0x0
Reg{MaxAllowed=4 ChkMtrs=1
ChkMsgRates=1
[Meters/Sec (Enabled: Min 0.00 Avg 0.17 Max 2.45 (Tot 28840.16
Samples 229013 SumAvg 0.13)]
[NotEnabled]
Timers=[3PMC=4 ACDSplt=61 AgntCls=30 AgntSt=240 BriHB=60
CfgRtry=900 StlBriMsg=10 SwtchTm=30 TG=60 StatMntr=28800
StatMntrInit=120]
SwitchTime=08/26/02 13:56:22
```

```
    NumActiveCalls=0 NumAgentsSeen=2
ProcessName=pim1 ShutdownType=1 Duplex=1 Side=0
    GeoTelBaseDir=c:\icm\lab1\PG1B RegistryBase=ICR\lab1\PG1B
DMPSystemID=1
    MDSConnections=1 MDSPIMHandle=33 MDSOPCHandle=1 PIMHeartBeatTime=-1
    CTIRestarts=0
    RoutingClientState=SHUTDOWN
    State=ACTIVE StateInitTime=08/27 10:06:55 (16.9 day)
Time stamp: 09/13/02 10:32:36
>>>>
```

See Also

For related information, see:

About Cisco Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the RTRTrace Utility

Use RTRTrace to set debug tracing on an ICM CallRouter. The additional tracing is output to the .EMS log files, viewable with the Log Collector utility.

Router Trace session can be started on either side of a Call Router, and tracing will start logging for both sides when tracing is enabled.

RTRTrace is for use with ICM Call Routers only.

Accessing RTRTrace

RTRTrace is an external GUI application present on all ICM nodes as part of that standard ICM installation. It cannot be run from within the Support Tools dashboard. Online documentation is available from within the RTRTrace GUI.

To access RTRTrace on an ICM Call Router, from a command line, enter `rtrtrace`.

See Also

For related information, see:

[About Cisco Common Utilities](#)

[Cisco TAC ICM Technical Notes on Cisco.com](#)

[How to Use the Log Collector Utility](#)

How to Use the RTTest Utility

Use the *RTTest* utility to interpret a Call Router's events and states without interruption to the running router processes. RTTest has several subroutines that allow you to view status, statistics, and similar information. You can also enable specific debug tracing in the Call Router. By running RTTest, you can quickly get a real time status of the entire ICM system.

For use with ICM Call Routers only, but can be run from any ICM component.

To Access RTTest from the Dashboard

To run RTTest from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > RTTest.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `nmstop` command is already implied. Do not enter it in the Arguments field.
4. If desired, in the Commands field, enter (or paste from a batch file) a group of commands to run in batch mode. When you do this:
 - Separate individual commands by a new line.
 - Do not enter input flags.
 - Do not enter an input file name in the arguments field.
5. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
6. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
7. Click the Run button.

To Access RTTest from a Command Line on a Node

From a command-line on an ICM Call Router, you can access RTTest from any location on the local physical drive. For example:


```
c:\>rttest
```

```
d:\temp>rttest
```

Note: You can access and run RTTest from other ICM components *provided* that the specified node in the command line argument (see next section) is an ICM Call Router.

Using RTTest

Syntax: `rttest [/f InputFile] [/system SystemName] [/cust Customer] [/node ICRNode] [/pipe OutputPipe] [/debug] [/stop] [/help] [/?]`

The command line options required to invoke RTTest are:

- `cust:` Customer where Customer is a 3, 4, or 5 letter acronym signifying the ICM customer instance.
- `node:` ICM node where ICM node is either `routera` or `routerb`, depending on the router RTTest to run.

At the RTTest prompt, type `status`. It will return the current state of each ICM central site process, ICM Peripheral Gateway server, third-party automatic call distributor (ACD) and Voice Response Unit (VRU) peripheral. For example:

```
c:\> rttest /cust cisco /node routera
rttest:
rttest: status
Router Version: Release 2.5 (service pack 2), Build 03134
Release Date:   12/23/98 13:30:08
Current Time:   03/17 16:00:42
Local Time:     03/17 11:00:42 (-5.0 hr)
Router Up:      02/21 01:01:45 (24.6 day)
Router Sync:    03/11 11:06:20 (6.2 day) (A->B)
```

Status Output

The three main sections to the `status` output are Process, Controller, and Peripheral.

Status Output: Process

The first section, labeled Process in the first column of the status output, shows the status of each ICM central site process. One ICM central site consists of an ICM call router and an ICM database logger. In most cases there will be two ICM central sites - `sideA` and `sideB` for redundancy.

Process:	LastStateChange:	LastHeartBeat:
A agi	- - -	--
A cic	- - -	--
A csfs	OK M- 03/06 11:10:20 (11.2 day)	--
A dba	OK MH 03/06 11:10:20 (11.2 day)	03/17 16:00:12 (30 sec)
A dbw	- - -	--
A lgr	OK MH 03/06 11:10:20 (11.2 day)	03/17 16:00:17 (25 sec)
A rcv	OK M- 03/06 11:10:20 (11.2 day)	--
A rtr	OK MH 03/06 11:10:20 (11.2 day)	03/17 16:00:15 (27 sec)
A rts	OK MH 03/06 11:10:20 (11.2 day)	03/17 16:00:19 (23 sec)
A tsyr	OK M- 03/06 11:10:20 (11.2 day)	--
B agi	- - -	--
B cic	- - -	--
B csfs	OK M- 03/11 11:08:34 (6.2 day)	--
B dba	OK MH 03/11 11:07:02 (6.2 day)	03/17 16:00:38 (4 sec)
B dbw	- - -	--
B lgr	OK MH 03/11 11:08:36 (6.2 day)	03/17 16:00:17 (25 sec)
B rcv	OK M- 03/11 11:08:35 (6.2 day)	--
B rtr	OK MH 03/11 11:07:03 (6.2 day)	03/17 16:00:15 (27 sec)
B rts	OK MH 03/11 11:07:02 (6.2 day)	03/17 16:00:29 (13 sec)
B tsyr	OK M- 03/11 11:07:02 (6.2 day)	--

First some general information is displayed such as Router version and build date. Then some statistics are displayed:

Current Time	This is Coordinated Universal Time (UTC). Most telecommunications equipment use UTC time as a common time reference.
Local Time	This is ICM local time, as determined by time zone setting on the Cisco ICM Call Router.
Router Up	This is how long the Cisco ICM Call Router function has been up and running.
Router Sync	This shows which side of the Cisco ICM call router last sent a state transfer to the other side.

Within the Status Process output, LastStateChange contains these fields:

OK	Signifies that the process is running fine.
M	Signifies the Cisco proprietary Message Delivery Service (MDS) protocol is used to keep the process synchronized.

H	Signifies that the process sends and receives internal heartbeat messages using the MDS protocol.
Date	Current date.
Time	Current local time.
Up-Time	In parenthesis is length of time the process has been in current state.
LastHeartBeat	If the process participates in sending and receiving MDS heartbeats, this value is the timestamp of the last heartbeat that was sent/received by the process.

Status Output: Controller

The second section, labeled Controller in the first column of the status output, shows the status of Cisco ICM peripheral gateway servers.

Controller is the name of the controller (ICM Peripheral Gateway) as defined in Configure initial cell rate (ICR).

Controller:	LastStateChange:	LastHeartBeat:
ATT_NIC_1,128	CFO 03/06 11:10:22 (11.2 day)	03/17 16:00:39 (3 sec)
ATT_NIC_2,129	CFO 03/11 11:07:05 (6.2 day)	03/17 16:00:34 (8 sec)
CA_PG9,9	CFO 03/17 04:42:31 (11.3 hr)	03/17 16:00:31 (11 sec)
FL_PG7,7	CFO 03/11 10:30:16 (6.2 day)	03/17 16:00:32 (10 sec)
GA_PG6,6	CFO 03/12 10:50:43 (5.2 day)	03/17 16:00:29 (13 sec)
IA_PG5,5	CFO 03/11 11:29:27 (6.1 day)	03/17 16:00:32 (10 sec)
NY_PG3,3	CFO 03/11 16:31:36 (5.9 day)	03/17 16:00:38 (4 sec)
TX_PG4,4	CFO 03/11 16:33:37 (5.9 day)	03/17 16:00:38 (4 sec)
VA_PG1,1	CFO 03/13 22:18:32 (3.7 day)	03/17 16:00:33 (9 sec)
VB_PG2,2	CFO 03/16 23:31:31 (16.4 hr)	03/17 16:00:32 (10 sec)

Within the Status Controller output, LastStateChange contains these fields:

C	Signifies that the ICM peripheral gateway server has successfully downloaded a configuration from the ICM call router.
F	Signifies that the ICM peripheral gateway is fully configured and the configuration is valid.
O	Signifies that the ICM peripheral gateway is online, and is communicating with the ICM call router.
Date	Current date.
Time	Current local time.
Up-Time	In parenthesis is length of time the process has been in current state.

Status Output: Peripheral

The third section labeled Peripheral in column 1, shows the status for third party peripherals such as ACD and VRU devices.

Peripheral is the name of the peripheral (ACD or VRU) as defined in Configure ICR.

Peripheral:	LastStateChange:	LastHeardFrom:
CA_PG9	COS 03/17 04:42:38 (11.3 hr)	03/17 16:00:40 (2 sec)
FL_PG7	COS 03/11 10:30:18 (6.2 day)	03/17 16:00:40 (2 sec)
GA_PG6	COS 03/16 06:21:18 (33.6 hr)	03/17 16:00:41 (1 sec)
IA_PG5	COS 03/11 11:29:30 (6.1 day)	03/17 16:00:40 (2 sec)
NY_PG3	COS 03/11 16:31:42 (5.9 day)	03/17 16:00:41 (1 sec)
TX_PG4	COS 03/11 16:37:53 (5.9 day)	03/17 16:00:34 (8 sec)
VA_PG1	COS 03/13 22:18:40 (3.7 day)	03/17 16:00:41 (1 sec)
VB_PG2	COS 03/16 23:31:33 (16.4 hr)	03/17 16:00:41 (1 sec)

Within the Status Peripheral output, LastStateChange contains these fields:

C	Signifies the peripheral is configured correctly to communicate with the ICM peripheral gateway.
O	Signifies the peripheral is online, for example, communications have been established with the ICM peripheral gateway.
S	Signifies that the peripheral is in service, for example, agent and call data are sent to the ICM peripheral gateway.
Date	Current date.
Time	Current local time.
Up-Time	In parenthesis is length of time the process has been in current state.
LastHeardFrom	Is the date, time, and length of time since the peripheral last sent valid data to the ICM peripheral gateway.

Parameter Descriptions

agent_status: Displays the states of agents and their groups.

agent_trace: Controls agent tracing in the router.

agi_lookup: Sends the router a application gateway lookup request.

checksum, check: Requests router to do a checksum.

cic_lookup: Sends the router a CIC lookup request.

cicr_time: Send ReportCICRTime message to the router.

closed_call: Sends a Closed call request to the router.

config: Enables config commands.

confignic: Sends a ConfigureNIC request to the router

configpd: Sends a ConfigurePD request to the router

configpg: Sends a ConfigurePG request to the router

`configrc`: Sends a ConfigureRC request to the router

`db_lookup`: Sends the router a `db_lookup` request.

`db_meters`: Displays meters about database accesses.

`dbw_status`: Display the dbworker status.

`debug_control`, `debug`: Controls various kinds of debugging output from router.

`deconfig`: Sends an message telling the router to de-configure.

`dump`: Requests router to do a internal state dump to a file.

`dump_adminscript_info`: Display Admin Script Information.

`dump_adminscript_runtimes`: Display Admin Script Run Times.

`dump_call_types`: Display the routers call type schedules.

`dump_datain`: Display the routers data input queue.

`dump_hash`: Display the routers internal hash statistics.

`dump_indirect_route_meters`: Display the routers indirect route meters.

`dump_locks`: Display the status of locks.

`dump_region`: Display the contents of a region.

`dump_struct`: Requests router to dump a structure.

`dump_timers`: Sends an message telling the router to display its timers.

`dump_vars`: Dumps info about router variables.

`echo`: Controls echoing of command lines

`enable_config`: Makes config commands visible.

`error_stop`: Controls setting of stop on error flag.

`exit`: Obsolete. Use `exit_router` or `quit`.

`exit_dbw`: Causes dbworker process to exit.

`exit_router`: Sends an message telling the router to exit.

`expression`, `expr`: Gives the router an expression to evaluate.

`get_config`: Causes the router to get config from logger.

`help`, `?`: Displays this help.

`mem_checkpoint`: Controls SmartHeap checkpointing.

`mem_meters`: Displays router memory meters.

`opi_data`: Send OPI data message to the router.

`quit, q`: Ends the program.

`read_file, read`: Directs command input to another input file

`rel_tranroute`: Simulate PG releasing a translation route.

`route_call`: Sends a route call request to the router.

`send_alarm`: Forces the router to send a message as an alarm.

`send_get_config`: Sends a GetConfigInd request to the router

`send_schedule`: Sends a schedule to the router.

`set_half_hour`: Sends a SetHalfHour request to the router

`startrc`: Sends a StartRC request to the router

`status`: Sends an message telling the router to display its status.

`stop_log`: Requests logmsg to stop logging messages.

`symbols`: Displays names of objects and symbols.

`test_edit_config`: Causes router to load config with the scripted interface.

`test_edit_script`: Causes router to test the script real time data interface.

`test_indirect_route`: Test request for ScriptIndirectRoute.

`test_route`: Analyzes routes for valid labels.

`test_update_script`: Causes router to test the script update interface.

`tuning_param, tune`: Controls tuning in the router.

`watch`: Sends watch expressions to the router.

Turning up ICM Call Router Tracing with RTTest

You can enable specific trace levels within RTTest by issuing the `debug` command, followed by one or more trace options. Respective trace entries can then be viewed in router logs.

For example, `debug /route` turns up tracing to show:

- Dialed Number (DN)
- Automatic Number Identification (ANI)
- Caller Entered Digits (CED), if any
- ICM routing label returned to the carrier network.

To see all possibilities for `/debug`, at the RTTest prompt, enter `debug /?` as shown below.

```
rttest: debug /? Usage: debug_control [/realtime] [/5minute] [/agent]
[/config] [/route] [/halfhour] [/rcmeter] [/expr] [/select] [/dupadd]
[/failpgerror] [/symbol] [/tranroute] [/datain] [/delivery] [/cic]
[/admin] [/pervarsumm] [/pervardetail] [/expform] [/vru] [/callq]
[/activepath] [/all] [/help] [/?]
```

All ICM processes write some default level of tracing to log files that can be viewed with the Log Collector utility.

- When specific trace levels are enabled, corresponding details are written to router log files in the logfile directory.
- Default individual log file size is 99k.
- Default aggregate log file size is 600k
- If router tracing is turned too high, individual log files will wrap very quickly, potentially within a minute if call volume is high. In that case, not much data can be captured because the time span is very small. To get around this, router log file capacities can be increased by altering a few Microsoft Windows NT registry settings. Make sure there is enough disk space available before increasing log file capacities.

To enter the Windows NT registry:

- From a command prompt, enter `regedt32`.
- After checking available disk space, the following two registry settings can be changed to allow for larger router log files.

Note: The values are displayed in hexadecimal by default. Click on the Decimal radio button to see the base 10 value.

```
\\.\software\geotel\icr\cisco\routera\ems\currentversion\library\proces-
ses\rtr\EMSAAllLogFilesMax
\\.\software\geotel\icr\cisco\routera\ems\currentversion\library\proces-
ses\rtr\EMSLogFileMax
```

The first parameter - `EMSAAllLogFilesMax` specifies the maximum amount of disk space the router will allocate for all log files combined.

The second parameter - `EMSLogFileMax` specifies the maximum size that the router will allocate to each log file. For example, if you set `EMSAAllLogFilesMax` to 20 mg, and `EMSLogFileMax` to 2 mg, the router will eventually create no more than 10 files, each being no more than 2 mg in size.

Turning Off Debug Tracing in RTTest

When you are finished viewing router logs, it is good practice to disable all tracing that was added for troubleshooting purposes. Do this using the `/noall` directive as shown below:

```
c:\icm\cd\ra\logfiles>rttest /cust cd /node routera
rttest: debug /noall
```

Ending an RTTest Session

It is very important that you quit from your RTTest session when finished. If too many RTTest sessions are left running in the background, system resources will be drained and call routing will be adversely affected.

To quit an RTTest session, enter: `rttest: quit`

See Also

For related information, see:

About Cisco Common Utilities

The Cisco ICM CTI Programmer's Guide

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the SS7NICTrace Utility

Use SS7NICTrace to view and set debug trace bits on an ICM SS7 NIC PG.

Accessing SS7NICTrace

SS7NICTrace is an external GUI application present on all ICM nodes as part of that standard ICM installation. It cannot be run from within the Support Tools dashboard.

To access SS7NICTrace on an SS& NIC PG, from a command line, enter `ss7nictrace`.

See Also

For related information, see:

[About Cisco Common Utilities](#)

How to Use the VRUTrace Utility

Use the *VRUTrace* utility to output tracing information from a Voice Response Unit (VRU) device, and to and from its PIM process. VRUTrace lets you capture and playback session data.

For use with ICM VRU PGs only.

To Access VRUTrace from the Dashboard

To run VRUTrace from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > VRUTrace.
3. In the Arguments field, enter desired arguments, as described in the Using section below. **Note:** When entering arguments, the `vrutrace` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

To Access VRUTrace from a Command Line on a Node

From a command-line on an ICM VRU PG, you must access VRUTrace from the `vrucap` directory located at: `<icm_root>\icm\<customer instance>\<pg#>`.

For example:

```
c:\icm\my_customer_instance\pg1a\vrucap> vrutrace
```

Using VRUTrace

Command Line Options

Syntax: vrutrace [ProcessName] [/if InputFile] [/o] [/of OutputFile] [/bd BeginDate(mm/dd/yy)] [/bt BeginTime(hh:mm:ss)] [/ed EndDate(mm/dd/yy)] [/et EndTime(hh:mm:ss)] [/all] [/prev] [/last] [/binary] [/debug] [/sleep] [/htbt] [/vrustat] [/help] [/?]

Examples

This example shows how you can obtain the output from a log file with information from April 29, 2003 until April 30, 2003 with binary data.

```
c:\icm\xyz\pg1a\vrucap\vrutrace pim1 /bd 04/29/2003 /ed 04/30/2003 /binary
```

This example shows how you can obtain output from a log file with information from the last time the process ran to the current date and time.

```
c:\icm\xyz\pg1a\vrucap\vrutrace pim1 /last
```

See Also

For related information, see:

About Cisco Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

Section V. Using Common Utilities

About Common Utilities

Common utilities are a set of third-party utilities, present on all or most Windows/ICM systems, that may be useful when troubleshooting ICM components. For the sake of convenience, the Support Tools Dashboard provides direct access to these utilities through its interface.

Note: This guide does not provide detailed documentation on using third-party tools. In all cases, extensive documentation is available from a variety of sources, including user guides, the Web, and from the tools themselves. For those third-party utilities on which Cisco-specific documentation does exist on <http://www.cisco.com>, links have been provided.

Accessing Common Utilities

Most common utilities are command-line based, and can be run from either the Support Tools Dashboard or from an individual node. Command-line access to individual nodes can be local or remote (via methods like TelNet, pcAnywhere, etc.).

Several common utilities are GUI-based. While menu commands for these utilities appear in the Support Tools Dashboard, they cannot in fact be launched from there. Access to these tools is limited to the individual node (either locally or via remote GUI such as pcAnywhere). These GUI-based third-party tools include:

- isql/w
- sqlw
- winmsd

Common Utilities Locations

Almost all Common Utilities are installed on all Support Tools nodes/Windows OSs. Several (see the table in the next section) may be particular to SQL Server installations.

Common Utilities at a Glance

The following Support Tools utilities are bundled third-party Windows, DOS, and Unix utilities. Most can be run both from within the Dashboard interface, or from an external command line.

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	No Dashboard or Command-Line Access:
Arp-a	Use to find the media access control address of a host on the same physical network.	All Support Tools nodes		
cat	Use the <i>CAT</i> utility to display, print, and combine files. Equivalent of the DOS type command.	All Support Tools nodes		
chmod	Use to change file permissions. Equivalent of the DOS attrib command.	All Support Tools nodes		
cp	Equivalent of the DOS copy command.	All Support Tools nodes		
df	Use to view disk free information.	All Support Tools nodes		
diff	Use to compare two text files or two directories.	All Support Tools nodes		
du	Use to view disk usage information.	All Support Tools nodes		
fgrep	Use to search for text patterns within a set of files.	All Support Tools nodes		
findstr	Use to find strings in logs or text files. Similar to UNIX's grep command. Useful when parsing log files or other text files to find a snippet of text.	All Support Tools nodes		
grep	Use to search for text patterns within a set of files.	All Support Tools nodes		
head	Use to view user-defined number of lines from the start of a file.	All Support Tools nodes		
IPConfig -all	Use to get host computer configuration information, including the IP address, subnet mask, and default gateway.	All Support Tools nodes		
isql/w	Interactive SQL for Windows. Use to query and update Microsoft SQL Server databases.	Nodes on which SQL Server is installed		X
ls	Use to view directory listings. Equivalent of the DOS dir command.	All Support Tools nodes		
mv	Use to move files and directories. Equivalent of the DOS move or rename command.	All Support Tools nodes		
NBTStat	Use to troubleshoot NetBIOS name resolution problems.	All Support Tools nodes		

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	No Dashboard or Command-Line Access:
Net Session	Use to view information about all current client sessions with the current system.	All Support Tools nodes		
NetStat	Use to view network information for the current system, including protocol statistics and current TCP/IP network connections.	All Support Tools nodes		
Net Statistics Server	Use to view statistics for the local Server service.	All Support Tools nodes		
Net Statistics Works	Use to view statistics for the local Workstation service.	All Support Tools nodes		
NSLookup	Use to get the IP address and fully qualified DNS of the current system's DNS name server.	All Support Tools nodes		
PathPing	A route tracing tool that combines features of the ping and tracert commands with additional information.	All Support Tools nodes		
ping	Low-level connectivity test. Use to verify end-to-end TCP/IP network infrastructure.	All Support Tools nodes		
pstat	Use to list all NT or Win2K processes, their process ID (PID), memory and cpu utilization, etc.	All Support Tools nodes		
rm	Use to delete files and directories. Equivalent of the DOS delete, recursive del subdirectories commands.	All Support Tools nodes		
Route-print	Use to view the contents of the network routing tables.	All Support Tools nodes		
shutdown	Use to shut down a remote host. When used from the Dashboard, shutdown will automatically restart the host after 60 seconds.	All Support Tools nodes	X	
sqlw	SQL Enterprise Manager. Use to monitor and update SQL Server operating parameters, expand databases, etc.	Nodes on which SQL Server is installed		X
stopshut	Use to stop a system from shutting down due to an application or operating system failure.	All Support Tools nodes		
strings	Use to find printable strings in a binary file.	All Support Tools nodes		
tail	Use to view user-defined number of lines from end of a file.	All Support Tools nodes		

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	No Dashboard or Command-Line Access:
touch	Use to change the creation date/time for any file.	All Support Tools nodes		
tracert	Related to ping command. Use to show each intermediary Hop in the TCP/IP network. Useful in troubleshooting LAN and WAN faults.	All Support Tools nodes		
wc	Use to view a count of characters, words, or lines in a file.	All Support Tools nodes		
which	Use to view the name of the first file encountered while traversing a path or directory tree.	All Support Tools nodes		
winmsd	Windows NT Diagnostics. Use to gather detailed configuration information about computers running Windows NT.	All Support Tools nodes		X

How to Use the Arp -a Utility

The Address Resolution Protocol (Arp) allows a host to find the media access control address of a host on the same physical network, given the IP address of the host. To make ARP efficient, each computer caches IP-to-media access control address mappings to eliminate repetitive Arp broadcast requests.

You can use the `arp -a` command to view the Arp table entries on a specified host. The `arp` command is useful for viewing the Arp cache and resolving address resolution problems.

To Access Arp -a from the Dashboard

To run Arp -a from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Arp -a.
3. Click the Run button.

Note: Because the Dashboard automatically sets the allowable commands for this tool, the Dashboard screen for this utility does not contain a command line input field.

Example

The following example shows sample output from the `arp -a` command.

```
Interface: 10.86.177.19 on Interface 0x1000003
  Internet Address      Physical Address      Type
  10.86.177.1          00-04-c1-34-df-98    dynamic
  10.86.177.10         00-d0-b7-ba-10-04    dynamic
  10.86.177.11         00-d0-b7-ba-10-d3    dynamic
  10.86.177.24         00-04-23-26-14-fe    dynamic
  10.86.177.71         00-d0-b7-ba-16-96    dynamic
```

See Also

For related information, see:

About Common Utilities

How to Use the CAT Utility

Use the *CAT* utility to display, print, and combine files.

To Access CAT from the Dashboard

To run CAT from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > CAT.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the *CAT* command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using CAT

Syntax

```
cat [ -usx? ] [ -v [-t] [-e] ] file1 ...
```

Command Line Options

The options for CAT are:

- u: output characters one at a time, not a line at a time.
- s: be silent about non-existent files.
- v: visible mode - display control characters as \wedge <char> (except for tabs, new-lines, and form-feeds) and chars with the 8th bit set as M-<char>.

-t: show tabs as ^I (only valid if -v given).

-e: show new-lines as \$ (only valid if -v given).

-x: expand tabs into the number of spaces specified by the TABS environment variable or into 8 spaces if TABS is undefined.

-n: no output - don't actually do any output, just do input. Equivalent to '> nul', except that on some platforms, the latter can be very slow (much slower than writing to an actual file.) This option is handy for seeing how long it takes to just read a set of files under varying operating conditions.

-?: Display program description.

Note: If the environment variable CAT exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the -v option, specify -v-.

Note: If the environment variable TABS exists, its value will determine the number of spaces used to expand tabs. Otherwise, tabs are expanded to 8 spaces.

See Also

For related information, see:

About Common Utilities

How to Use the CHMOD Utility

Use the *CHMOD* utility to set file permissions.

To Access CHMOD from the Dashboard

To run CHMOD from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > CHMOD.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the *CHMOD* command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using CHMOD

Syntax

```
chmod <mode> file1 ...
```

Command Line Options

Mode is any combination of {+|-|=} {w|s|h|m} where:

- + adds an attribute to a file
- removes an attribute from a file
- = gives a file just the attributes specified

w specifies whether the file should be writable or not

s specifies whether the file should be a system file or not

h specifies whether the file should be a hidden file or not

m specifies whether the file should be marked modified or not

-?: Display program description.

Note: If the environment variable CHMOD exists, its value is used to establish default options.

See Also

For related information, see:

About Common Utilities

How to Use the CP Utility

Use the *CP* utility to copy files.

To Access CP from the Dashboard

To run CP from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > CP.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `CP` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using CP

Syntax

```
cp [-ifprv?] file1 file2
```

```
cp [-ifprvd?] file1 fileN directory
```

If the last file is a directory, each file is copied into a file in the destination directory with the same name.

If only two files are specified, file1 is copied to file2.

If more than two files are specified, the last file must be a directory.

Command Line Options

The options for CP are:

-i: ask for confirmation before overwriting each destination file.

-f: copy the file, don't ask for confirmation even if it will overwrite a file marked Read-Only.

-p: preserve the source attributes and last modified date and time for the destination file. This is the default. To turn it off use **-p-**.

-r: recursively copy all of the files and subdirectories specified by the filename, including the named directory.

-v: print out status as the cp progresses.

-d: preserve the directory structure of the specified files (which must all be specified with relative paths) when placing them in the specified directory. For example, '`cp -d foo\bar\baz otherdir`' will make the directories `otherdir\foo` and `otherdir\bar` if necessary, then copy `foo\bar\baz` to `otherdir\foo\bar\baz`.

-?: Display program description.

Note: If the environment variable CP exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the **-i** option, specify **-i-**.

Note: The order of options **f** and **i** is significant: the one specified last determines whether a confirmation is requested.

See Also

For related information, see:

About Common Utilities

How to Use the DF Utility

Use the *DF* utility to view information on file system disk space usage.

To Access DF from the Dashboard

To run DF from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > DF.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the *DF* command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using DF

Syntax

```
df [ -a | drive_1 [ drive_2 ... ] | -? ]
```

If DF is not given an argument, it will print the free space information for the default drive. Otherwise, unless the *-a* option is specified, DF prints the free space information for each of the supplied arguments.

Example: `df a: b:` prints the available free space for drives a: and b:.

Command Line Options

The options for DF are:

-a: Print free space for all locatable drives.

-?: Display program description.

Note: If the environment variable DF exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the -a option, specify -a-.

Note: Option -a can be specified twice to restore normal operation, namely, to undo the effect of specifying option -a in the environment variable.

See Also

For related information, see:

About Common Utilities

How to Use the Diff Utility

Use the *Diff* utility to find differences between two text files.

To Access Diff from the Dashboard

To run Diff from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > Diff.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `Diff` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Diff

Syntax

```
diff [ -rebidtv? ] [-O <colors>] oldfile newfile
```

If `diff` is given two directories, it will diff each of the files in those directories.

If `diff` is given two binary files, it will compare them and issue a report if they are different.

If `diff` is given the `-r` option, it will recursively diff each of the files in each subdirectory.

If `diff` is given the `-e` option, it will output a script suitable for feeding into EDLIN to update oldfile to create newfile.

Otherwise, `diff` outputs lines of the form:

n1 a n3,n4

n1,n2 d n3

n1,n2 c n3,n4

where 'a' means: add lines n3 thru n4 from newfile at location n1 in newfile; 'd' means: delete lines n1 thru n2 from newfile, next line is n3 in newfile; and 'c' means: change lines n1 thru n2 in newfile to lines n3 thru n4 in newfile. Identical pairs where n1=n2 or n3=n4 are abbreviated as a single number. Following each of the [adc] lines comes each of the lines effected. Lines preceded by '<' are from oldfile, those preceded by '>' are from newfile.

Command Line Options

The options for Diff are:

-b: Ignore trailing whitespace and treat sequences of embedded whitespace as being equal.

-i: Ignore leading whitespace.

-d: Treat files as if they were binary, only reporting if they are different.

-t: Treat files as if they were text, even if default autodetection claims they are binary.

-v: Report on each file that it processes, not just the ones with differences.

-o: Colorize output with the default colors (black, white, light cyan, and yellow), just as '-O Ofbe' would.

-O <colors>: Colorize output with the specified single-character hex colors: background, normal text, old text, and new text. The colors are: black = 0, blue = 1, green = 2, cyan = 3, red = 4, magenta = 5, brown = 6, lightgray = 7, darkgray = 8, lightblue = 9, lightgreen = A, lightcyan = B, lightred = C, lightmagenta = D, yellow = E, white = F.

-?: Display program description.

Note: If the environment variable DIFF exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the -b option, specify -b-.

See Also

For related information, see:

About Common Utilities

How to Use the DU Utility

Use the *DU* utility to view the sizes of each directory specified on the command line and the sizes of each subdirectory under those.

To Access DU from the Dashboard

To run DU from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > DU.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `DU` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using DU

Syntax

```
du [ -asM? ] [ names ]
```

If no names are specified on the command line, the size of the current directory is given. Sizes given show the total number of bytes in all of the files within and below the named directory and the amount of disk space used by the files in and below the named directory. The disk space is displayed in Kbytes (1024 bytes = 1 Kbyte)

Command Line Options

The options for DU are:

-a: also display the size of each file.

-s: only display the grand totals for the names on the command line.

-M: show file names in their OS-provided mixed-case forms, rather than lower-casing them. Note however that most file systems don't allow files with names differing only by case, and will ignore case differences when looking for a file.

-?: Display program description.

Note: If the environment variable DU exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the -a option, specify -a-.

See Also

For related information, see:

About Common Utilities

How to Use the FGrep Utility

Use the *FGrep* utility to perform a fast search for text patterns through text files.

To Access FGrep from the Dashboard

To run FGrep from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > FGrep.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `FGrep` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using FGrep

Syntax

```
fgrep [ -mvclxinfse? ] <pattern> file1 ...
```

If no names are specified on the command line, the size of the current directory is given. Sizes given show the total number of bytes in all of the files within and below the named directory and the amount of disk space used by the files in and below the named directory. The disk space is displayed in Kbytes (1024 bytes = 1 Kbyte)

Command Line Options

The options for FGrep are:

`-m`: print the lines that match the pattern (DEFAULT).

-v: print the lines that DON'T match the pattern.

-c: print a count of the number of matching lines in each file.

-l: print the name of each file that has a match.

Note: Options `m`, `v`, `c` and `l` are exclusive. Only one of these may be selected.

-x: the pattern must match the entire line to succeed.

-i: ignore the case of each character while matching.

-n: print the line number that the match occurred on.

-f: print the full path name of the file when printing filenames.

-s: do not report errors encountered while opening or reading files.

-e: use the next argument as the pattern to search for. This is useful if the pattern begins with a `-` or `/` character.

-?: Display program description.

Note: If the environment variable `FGrep` exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the `-i` option, specify `-i-`.

See Also

For related information, see:

About Common Utilities

How to Use the Grep Utility

How to Use the Findstr Utility

Use the *Findstr* utility to find strings in logs or text files. Findstr is useful when parsing log files or other text files for a snippet of text.

To Access Findstr from the Dashboard

To run Findstr from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > Findstr.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `Findstr` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Findstr

Syntax

```
FINDSTR [/B] [/E] [/L] [/R] [/S] [/I] [/X] [/V] [/N] [/M] [/O] [/P]
[/F:file] [/C:string] [/G:file] [/D:dir list] [/A:color attributes]
[strings] [[drive:]][path]filename[ ...]
```

If no names are specified on the command line, the size of the current directory is given. Sizes given show the total number of bytes in all of the files within and below the named directory and the amount of disk space used by the files in and below the named directory. The disk space is displayed in Kbytes (1024 bytes = 1 Kbyte)

Command Line Options

The options for Findstr are:

`/B`: Matches pattern if at the beginning of a line.

`/E`: Matches pattern if at the end of a line.

`/L`: Uses search strings literally.

`/R`: Uses search strings as regular expressions.

`/S`: Searches for matching files in the current directory and all subdirectories.

`/I`: Specifies that the search is not to be case-sensitive.

`/X`: Prints lines that match exactly.

`/V`: Prints only lines that do not contain a match.

`/N`: Prints the line number before each line that matches.

`/M`: Prints only the filename if a file contains a match.

`/O`: Prints character offset before each matching line.

`/P`: Skip files with non-printable characters

`/A:attr`: Specifies color attribute with two hex digits. See "color /?"

`/F:file`: Reads file list from the specified file (/ stands for console).

`/C:string`: Uses specified string as a literal search string.

`/G:file`: Gets search strings from the specified file (/ stands for console).

`/D:dir`: Search a semicolon delimited list of directories

`strings`: Text to be searched for.

`[drive:] [path] filename`: Specifies a file or files to search.

`-?`: Display program description.

Use spaces to separate multiple search strings unless the argument is prefixed with `/C`. For example, `'FINDSTR "hello there" x.y'` searches for "hello" or "there" in file `x.y`. `'FINDSTR /C:"hello there" x.y'` searches for "hello there" in file `x.y`.

Regular expression reference:

`.` Wildcard: any character

`*` Repeat: zero or more occurrence of previous character or class

`^` Line position: beginning of line

`$` Line position: end of line

[class] Character class: any one character in set

[^class] Inverse class: any one character not in set

[x-y] Range: any characters within the specified range

\x Escape: literal use of metacharacter x

\<xyz Word position: beginning of word

xyz\> Word position: end of word

See Also

For related information, see:

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the Grep Utility

Use the *Grep* utility to perform a regular expression search for text patterns through text files.

To Access Grep from the Dashboard

To run Grep from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > Grep.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `Grep` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Grep

Syntax

```
grep [ -mvclinfse? ] <pattern> file1 ...
```

Command Line Options

The options for Grep are:

- m: print the lines that match the pattern (DEFAULT).
- v: print the lines that DON'T match the pattern.
- c: print a count of the number of matching lines in each file.

-l: print the name of each file that has a match.

Note: Options `m`, `v`, `c` and `l` are exclusive. Only one of these may be selected.

-i: ignore the case of each character while matching.

-n: print the line number that the match occurred on.

-f: print the full pathname of the file when printing filenames.

-s: do not report errors encountered while opening or reading files.

-e: use the next argument as the pattern to search for. This is useful if the pattern begins with a `-` or `/` character.

-?: Display program description.

The specified pattern is treated as a regular expression -- if all you need is a literal expression, you may want to use `fgrep` instead. The actual regular expression syntax is fairly standard, but escape sequence issues can complicate it. Here's an example of the 'pristine' syntax seen internally:

```
^a.b+c*(d|e)?[^\f]*"$
```

means "in beginning-of-line context, match 'a', then an arbitrary character, then one or more 'b's, then zero or more 'c's, then one of 'd', 'e', or nothing, then any character but 'f', then a literal '*' character, then a double-quote character, in end-of-line context."

The first added complication is the Picnix command-line argument parser, which has special semantics for `"`, `'`, ```, `@`, `$`, and `^`. So the argument as seen by this parser would have to be:

```
^^a.b+c*(d|e)?[^^f]*^^"$
```

The second added complication is that you may be executing `grep` through an additional shell, such as the WinNT `cmd.exe`, which has special semantics for `|`, `^`, `"`, and `\`, and for which you should actually put this on the command line or in your batch file:

```
"^^a.b+c*(d|e)?[^^f]*^^"$
```

Or you may be executing `grep` through a build tool like `pmake.exe`, for which you'd have to say:

```
"^^^^a.b+c*(d|e)?[^^^^f]*^^"$
```

Note: If the environment variable `Grep` exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus `'-'` sign. For example, to turn off the `-i` option, specify `-i-`.

See Also

For related information, see:

[How to Use the FGrep Utility](#)

How to Use the Head Utility

Use the *Head* utility to view a user-defined number of lines from the start of a file.

To Access Head from the Dashboard

To run Head from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > Head.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `Head` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Head

Head displays the first <count> lines of each of the files listed on the line. If no files are listed, display the first <count> lines of the standard input.

<count> defaults to 10.

Syntax

```
head [-<count> | -n <count>] [-vx?] [ file1 ... ]
```

Command Line Options

The options for Head are:

-<count> or -n <count>: specifies the number of lines to display.

-v: Print headers giving file names.

-x: Expand tabs into the number of spaces specified by the TABS environment variable or into 8 spaces if TABS is undefined.

-?: Display program description.

Note: If the environment variable Head exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the -v option, specify -v-.

Note: If the environment variable TABS exists, its value will determine the number of spaces used to expand tabs. Otherwise, tabs are expanded to 8 spaces.

See Also

For related information, see:

About Common Utilities

How to Use the FHead Utility

How to Use the IPConfig -all Utility

When you troubleshoot a TCP/IP networking problem, begin by checking the TCP/IP configuration on the computer that is experiencing the problem.

Use the *IPConfig -all* command to get host computer configuration information, including the IP address, subnet mask, and default gateway. When you use the IPConfig command with the -all option, a detailed configuration report is produced for all interfaces, including any configured serial ports.

To Access IPConfig -all from the Dashboard

To run IPConfig -all from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > IPConfig -all.
3. Click the Run button.

Note: Because the Dashboard automatically sets the allowable commands for this tool, the Dashboard screen for this utility does not contain a command line input field.

Example

The following example shows sample output from the IPConfig -all command.

How to Use the ISQL/W Utility

Use the ISQL/W utility to query and update Microsoft SQL Server databases.

Accessing ISQL/W

ISQL/W is a Microsoft external GUI application present on systems with Microsoft SQL Server installed. It cannot be run from within the Support Tools dashboard.

To access ISQL/W:

1. On the Windows Start bar, click Start > Run.
2. In the Open field, enter `isqlw`.
3. Click OK.

See Also

For related information, see:

[About Common Utilities](#)

[Cisco TAC ICM Technical Notes on Cisco.com](#)

[Using the ISQL Utility \(Microsoft.com\)](#)

How to Use the LS Utility

Use the *LS* utility to view directory listings.

To Access LS from the Dashboard

To run LS from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > LS.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `LS` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using LS

Syntax

```
ls [ -cxllmfetsurRpadoM? ] [ file1 ... ]
```

Command Line Options

The options for LS are:

- c: display the names of the files sorted down each column.
- x: display the names of the files sorted across the row.
- 1: display the names of the files in a single column.

- l: display a long listing with more information about each file.
- m: display the names of the files separated by commas.
- f: sort the files by filename.
- e: sort the files first by the extension and then by the filename.
- t: sort the files by the date and time that the files were last written.
- s: sort the files by the size (in bytes).
- u: do not sort the files.
- r: reverse the order of the sort selected.
- R: recursively descend down each subdirectory.
- p: put a slash (/ or \) after each directory.
- a: list all files, including HIDDEN files.
- d: when a directory is specified on the command line, display it as an individual item rather than displaying its contents.
- o: when a high-bit (greater than 127) character in a filename is displayed, assume the name is in the Windows/ANSI character set and convert it to the DOS/OEM character set, just like 'dir' does. This means that for commonly-used accent characters in Windows filenames, they'll look the same in a OEM character-set-based command prompt. Note however that this may result in output which obscures actual distinctions, since the conversion is not always reversible. For example, four accented Windows/ANSI versions of 'A' (0xc0 - 0xc4) all map to 'A' under this conversion. Also note that ls will only ever show single-byte filenames, even if the underlying file system has Unicode filenames -- when dealing with a file with Unicode characters in its name, the Picnix utilities will try to use the OS-provided 8.3 short filename instead.
- M: show file names in their OS-provided mixed-case forms, rather than lower-casing them. Note however that most file systems don't allow files with names differing only by case, and will ignore case differences when looking for a file.
- ?: Display program description.

Note: If the environment variable LS exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the -p option, specify -p-.

See Also

For related information, see:

[About Common Utilities](#)

How to Use the MV Utility

Use the *MV* utility to move files and directories.

To Access MV from the Dashboard

To run MV from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco CommonTooMV > MV.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the *MV* command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using MV

Syntax

```
mv [-firv?] file1 file2
```

```
mv [-firv?] file1 ... fileN directory
```

If the last file is a directory, each file is moved into a file in the destination directory with the same name.

If only two files are specified, file1 is moved to file2.

If more than two files are specified, the last file must be a directory.

Command Line Options

The options for MV are:

`-i`: ask for confirmation before overwriting each destination file.

`-f`: move the file, don't ask for confirmation even if it will overwrite a file marked Read-Only.

`-r`: recursively move all of the files and subdirectories specified by the filename, including the named directory.

`-v`: print out status as the mv progresses.

`-?`: Display program description.

Note: If the environment variable MV exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the `-i` option, specify `-i-`.

Note: The order of options `f` and `i` is significant: the one specified last determines whether a confirmation is requested.

Note: If the environment variable MV exists, its value is used to establish default options.

See Also

For related information, see:

About Common Utilities

How to Use the NBTStat Utility

Use the *NBTStat* utility to troubleshoot NetBIOS name resolution problems. NetBIOS over TCP/IP (NetBT) resolves NetBIOS names to IP addresses. TCP/IP provides many options for NetBIOS name resolution, including local cache lookup, WINS server query, broadcast, DNS server query, and Lmhosts and Hosts file lookup.

To Access NBTStat from the Dashboard

To run NBTStat from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > NBTStat.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using NBTStat

Syntax

```
nbtstat [ [-a RemoteName] [-A IP address] [-c] [-n] [-r] [-R] [-RR] [-s] [-S] [interval] ]
```

Command Line Options

The options for NBTStat are:

- a (adapter status) Lists the remote machine's name table given its name
- A (Adapter status) Lists the remote machine's name table given its IP address.
- c (cache) Lists NBT's cache of remote [machine] names and their IP addresses

- n (names) Lists local NetBIOS names.
 - r (resolved) Lists names resolved by broadcast and via WINS
 - R (Reload) Purges and reloads the remote cache name table
 - S (Sessions) Lists sessions table with the destination IP addresses
 - s (sessions) Lists sessions table converting destination IP addresses to computer NETBIOS names.
 - RR (ReleaseRefresh) Sends Name Release packets to WINS and then, starts Refresh
- RemoteName Remote host machine name.
- IP address Dotted decimal representation of the IP address.
- interval Redisplays selected statistics, pausing interval seconds between each display. Press Ctrl+C to stop redisplaying statistics.

See Also

For related information, see:

About Common Utilities

How to Use the Net Session Utility

Use the *Net Session* utility to view information about all current client sessions with the current system. Net Session displays the computer names and user names of users on a server, to see if users have files open, and to see how long each user's session has been idle.

To Access Net Session from the Dashboard

To run Net Session from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Net Session.
3. Click the Run button.

Note: Because the Dashboard automatically sets the allowable commands for this tool, the Dashboard screen for this utility does not contain a command line input field.

See Also

For related information, see:

About Common Utilities

Microsoft Tech Net (Microsoft.com)

How to Use the Net Statistics Server Utility

Use the *Net Statistics Works* utility to view statistics for the local Server service.

To Access Net Statistics Server from the Dashboard

To run Net Statistics Server from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Net Statistics Server.
3. Click the Run button.

Note: Because the Dashboard automatically sets the allowable commands for this tool, the Dashboard screen for this utility does not contain a command line input field.

Example

The following example shows sample output from the Net Statistics Server utility.

```

Server Statistics for \
Statistics since 7/16/2003 11:06 AM

Sessions accepted                1
Sessions timed-out               0
Sessions errored-out             0

Kilobytes sent                   14
Kilobytes received               13

Mean response time (msec)       0

System errors                   0
Permission violations            0
Password violations              0

Files accessed                   2
Communication devices accessed  0
Print jobs spooled              0

Times buffers exhausted

    Big buffers                   0
    Request buffers                0

```

See Also

For related information, see:

About Common Utilities

How to Use the Net Statistics Workstation Utility

Use the *Net Statistics Workstation* utility to view statistics for the local Workstation service.

To Access Net Statistics Workstation from the Dashboard

To run Net Statistics Workstation from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Net Statistics Workstation.
3. Click the Run button.

Note: Because the Dashboard automatically sets the allowable commands for this tool, the Dashboard screen for this utility does not contain a command line input field.

Example

The following example shows sample output from the Net Statistics Workstation utility.

Workstation Statistics for \\

Statistics since 7/16/2003 11:06 AM

Bytes received	255226
Server Message Blocks (SMBs) received	1189
Bytes transmitted	301087
Server Message Blocks (SMBs) transmitted	1183
Read operations	10
Write operations	0
Raw reads denied	0
Raw writes denied	0
Network errors	0
Connections made	139
Reconnections made	0
Server disconnects	0
Sessions started	0
Hung sessions	0
Failed sessions	0
Failed operations	0
Use count	148
Failed use count	0

The command completed successfully.

See Also

For related information, see:

About Common Utilities

How to Use the NetStat Utility

Use the *NetStat* utility to view network information for the target host, including protocol statistics and current TCP/IP network connections.

To Access NetStat from the Dashboard

To run NetStat from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > NetStat.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `NetStat` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using NetStat

Syntax

```
NetStat [-a] [-e] [-n] [-s] [-p proto] [-r] [interval]
```

Command Line Options

The options for NetStat are:

- a Displays all connections and listening ports.
- e Displays Ethernet statistics. This may be combined with the -s option.
- n Displays addresses and port numbers in numerical form.

`-p proto` Shows connections for the protocol specified by `proto`; `proto` may be TCP or UDP. If used with the `-s` option to display per-protocol statistics, `proto` may be TCP, UDP, or IP.

`-r` Displays the routing table.

`-s` Displays per-protocol statistics. By default, statistics are shown for TCP, UDP and IP; the `-p` option may be used to specify a subset of the default.

`interval` Redisplays selected statistics, pausing `interval` seconds between each display. Press CTRL+C to stop redisplaying statistics. If omitted, `netstat` will print the current configuration information once.

`-?:` Display program description.

See Also

For related information, see:

About Common Utilities

How to Use the NSLookup Utility

Use the *NSLookup* utility to get information on the current or specified system's DNS name server.

To Access NSLookup from the Dashboard

To run NSLookup from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > NSLookup.
3. enter the IP address or DNS of the system you want to query--OR--to get DNS name server information for the current system, leave the Arguments field blank.
Note: When entering arguments, the `NetStat` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. **Note:** For this utility, you may want to adjust this default number downwards; otherwise, it will not return a value until the command has timed out.
6. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
7. Click the Run button.

Using NSLookup

Syntax

```
nslookup ip_address or hostname
```

See Also

For related information, see:

About Common Utilities

How to Use the PathPing Utility

Use the *PathPing* utility as a route tracing tool that combines features of the Ping and Tracert utilities with additional information that neither of those tools provides (for example QOS testing).

To Access PathPing from the Dashboard

To run PathPing from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > PathPing.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `strings` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
7. Click the Run button.

Using PathPing

Syntax

```
pathping [-n] [-h maximum_hops] [-g host-list] [-p period] [-q num_queries] [-w timeout] [-t] [-R] [-r target_name]
```

Command Line Options

The options for PathPing are:

-n: Do not resolve addresses to hostnames.

- h `maximum_hops`: Maximum number of hops to search for target.
- g `host-list`: Loose source route along host-list.
- p `period`: Wait period milliseconds between pings.
- w `timeout`: Wait timeout milliseconds for each reply.
- T: Test connectivity to each hop with Layer-2 priority tags.
- R: Test if each hop is RSVP aware.

See Also

For related information, see:

About Common Utilities

How to Use the Ping Utility

Use the *Ping* utility to verify IP-level connectivity. When troubleshooting, you can use Ping to send an ICMP echo request to a target host name or IP address. Use Ping whenever you need to verify that a host computer can connect to the TCP/IP network and network resources. You can also use Ping to isolate network hardware problems and incompatible configurations.

To Access Ping from the Dashboard

To run Ping from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Ping.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Ping

Syntax

```
ping [-t] [-a] [-n count] [-l size] [-f] [-i TTL] [-v TOS] [-r count] [-s count] [[-j host-list] | [-k host-list]] [-w timeout] destination-list
```

Command Line Options

The options for Ping are:

-t: Ping the specified host until stopped. To see statistics and continue - type Control-Break; To stop - type Control-C.

-a: Resolve addresses to hostnames.

- n count: Number of echo requests to send.
- l sizeSend: buffer size.
- f set: Don't Fragment flag in packet.
- i TTL: Time To Live.
- v TOS: Type Of Service.
- r count: Record route for count hops.
- s count: Timestamp for count hops.
- j host-list: Loose source route along host-list.
- k host-list: Strict source route along host-list.
- w timeout: Timeout in milliseconds to wait for each reply.

See Also

For related information, see:

[About Common Utilities](#)

How to Use the PStat Utility

Use the *PStat* utility to list all the processes running on a Microsoft Windows NT Workstation, including ICM processes. You can use the PStat command to verify that an ICM process was not properly exited and is still running.

Note: You can also achieve this using the Support Tools Processes utility.

To Access PStat from the Dashboard

To run PStat from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > PStat.
3. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
4. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
5. Click the Run button.

Note: Because the Dashboard automatically sets the allowable commands for this tool, the Dashboard screen for this utility does not contain a command line input field.

Example

The following example shows sample output from the PStat command.

>pstat | findstr /i pid

User	Time	Kernel Time	Ws	Faults	Commit	Pri	Hnd	Thd	Pid	Name
pid: 0	pri: 0	Hnd: 0	Pf:	1	Ws:	16K	Idle	Process		
pid: 2	pri: 8	Hnd: 502	Pf:	8961	Ws:	200K	System			
pid: 17	pri: 11	Hnd: 30	Pf:	2301	Ws:	0K	smss.exe			
pid: 1f	pri: 13	Hnd: 698	Pf:	49103	Ws:	3144K	CSRSS.EXE			
pid: 25	pri: 13	Hnd: 59	Pf:	11625	Ws:	1504K	WINLOGON.EXE			
pid: 2d	pri: 9	Hnd: 267	Pf:	335153	Ws:	4216K	SERVICES.EXE			
pid: 30	pri: 9	Hnd: 114	Pf:	10471	Ws:	1572K	LSASS.EXE			
pid: 48	pri: 8	Hnd: 58	Pf:	3267	Ws:	204K	SPOOLSS.EXE			
pid: 54	pri: 8	Hnd: 197	Pf:	30864	Ws:	1852K	AWHOST32.EXE			
pid: 74	pri: 8	Hnd: 63	Pf:	5361	Ws:	220K	RPCSS.EXE			
pid: 82	pri: 8	Hnd: 28	Pf:	3954	Ws:	232K	ATSVC.EXE			
pid: 89	pri: 8	Hnd: 20	Pf:	485	Ws:	24K	SYSDOWN.EXE			
pid: 92	pri: 8	Hnd: 37	Pf:	20105	Ws:	84K	PSTORES.EXE			
pid: 182	pri: 8	Hnd: 74	Pf:	3597	Ws:	896K	TELNETD.EXE			
pid: fe	pri: 8	Hnd: 154	Pf:	1143	Ws:	2648K	Nodemans.exe			
pid: 195	pri: 13	Hnd: 32	Pf:	431	Ws:	40K	Nmm.exe			
pid: db	pri: 13	Hnd: 1759	Pf:	13934	Ws:	4588K	CCAGENT.exe			
pid: 1e4	pri: 13	Hnd: 87	Pf:	24538	Ws:	2104K	dbagent.exe			
pid: 134	pri: 13	Hnd: 101	Pf:	4337	Ws:	1768K	mcinic.exe			
pid: 1cd	pri: 13	Hnd: 431	Pf:	41132	Ws:	3092K	MDSPROC.exe			
pid: 13f	pri: 13	Hnd: 97	Pf:	667348	Ws:	24756K	router.exe			
pid: f0	pri: 13	Hnd: 288	Pf:	191433	Ws:	3220K	Rtsvr.exe			
pid: 135	pri: 13	Hnd: 150	Pf:	3472	Ws:	1348K	TESTSYNC.exe			
pid: 168	pri: 8	Hnd: 44	Pf:	1017	Ws:	200K	InspectService.			
pid: 146	pri: 8	Hnd: 29	Pf:	457	Ws:	52K	TRANSCMD.EXE			
pid: b8	pri: 8	Hnd: 25	Pf:	491	Ws:	44K	CMD.EXE			
pid: 109	pri: 8	Hnd: 29	Pf:	421	Ws:	52K	TRANSCMD.EXE			
pid: 64	pri: 8	Hnd: 25	Pf:	376	Ws:	44K	CMD.EXE			
pid: 12a	pri: 8	Hnd: 29	Pf:	427	Ws:	52K	TRANSCMD.EXE			
pid: 1ba	pri: 8	Hnd: 26	Pf:	378	Ws:	44K	CMD.EXE			
pid: e7	pri: 8	Hnd: 21	Pf:	471	Ws:	44K	Rttest.exe			
pid: 18b	pri: 8	Hnd: 29	Pf:	387	Ws:	1524K	TRANSCMD.EXE			
pid: 235	pri: 8	Hnd: 27	Pf:	332	Ws:	1260K	CMD.EXE			
pid: 1d3	pri: 8	Hnd: 20	Pf:	229	Ws:	912K	PSTAT.EXE			

See Also

For related information, see:

About Common Utilities

How to Use the Processes Screen

How to Use the RM Utility

Use the *RM* utility to delete files and directories.

To Access RM from the Dashboard

To run RM from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > RM.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `RM` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using RM

Syntax

```
rm [ -firv? ] file ...
```

Command Line Options

The options for RM are:

- i: ask for confirmation before removing each file.
- f: remove the file, don't ask for confirmation even if it is marked Read-Only.
- r: recursively delete all of the files and subdirectories specified by the filename, including the named directory.

-v: print out the name of each file that is removed.

-?: Display program description.

Note: If the environment variable RM exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the -i option, specify -i-.

See Also

For related information, see:

About Common Utilities

How to Use the Route -print Utility

Use the *Route -print* utility to view the contents of the network routing tables.

To Access Route -print from the Dashboard

To run Route -print from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Route -print.
3. Click the Run button.

Note: Because the Dashboard automatically sets the allowable commands for this tool, the Dashboard screen for this utility does not contain a command line input field.

Example

The following example shows sample output from the Route -print command.

```
=====
Interface List
Ox1 ..... MS TCP Loopback interface
Ox1000003 ...00 10 b5 48 17 7e ..... NDIS 5.0 driver
=====

Active Routes:
Network Destination        Netmask          Gateway           Interface        Metric
0.0.0.0                    0.0.0.0          10.86.177.1       10.86.177.19     1
10.86.177.0                255.255.255.0    10.86.177.19      10.86.177.19     1
10.86.177.19              255.255.255.255  127.0.0.1         127.0.0.1        1
10.255.255.255            255.255.255.255  10.86.177.19      10.86.177.19     1
127.0.0.0                  255.0.0.0        127.0.0.1         127.0.0.1        1
224.0.0.0                  224.0.0.0        10.86.177.19      10.86.177.19     1
255.255.255.255          255.255.255.255  10.86.177.19      10.86.177.19     1
Default Gateway:          10.86.177.1
=====

Persistent Routes:
None
```

See Also

For related information, see:

About Common Utilities

How to Use the Shutdown Tool Utility

Use the *Shutdown Tool* utility to stop and restart a remote host after a 60 second grace period.

To Access Shutdown from the Dashboard

To run Shutdown from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > Shutdown Tool.
3. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
4. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
5. Click the Run button.

Note: Because the Dashboard automatically sets the allowable commands for this tool, the Dashboard screen for this utility does not contain a command line input field.

See Also

For related information, see:

About Common Utilities

How to Use the SQLEW Utility

Use the *SQLEW* utility to monitor and update SQL Server operating parameters, expand databases, and so on.

Accessing SQLEW

SQLEW is a Microsoft external GUI application present on all Windows 2000 installations. It cannot be run from within the Support Tools dashboard.

To access SQLEW:

1. On the Windows Start bar, click Start > Run.
2. In the Open field, enter `sqllew`.
3. Click OK.

See Also

For related information, see:

[About Common Utilities](#)

[Cisco TAC ICM Technical Notes on Cisco.com](#)

[Microsoft Knowledge Base Search](#)

[How to Create a SQLEW Report \(Microsoft.com\)](#)

[How to Use SQLeWP.EXE \(Microsoft.com\)](#)

How to Use the Stopshut Utility

Use the *Stopshut* utility to stop a system from shutting down due to an application or operating system failure.

To Access Stopshut from the Dashboard

To run Stopshut from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Common Tools > Stopshut.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `stopshut` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Stopshut

Syntax

Syntax: `c:\>stopshut`

See Also

For related information, see:

About Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the Strings Utility

Use the *Strings* utility to print ASCII strings embedded in binary files.

To Access Strings from the Dashboard

To run Strings from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Strings.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `strings` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Strings

Strings displays printable ASCII strings in the binary files specified on the command line. If no files are given, it displays printable ASCII strings in the standard input.

Syntax

```
strings [ -? ] [ -t <n> ] [ -n <n> ] [ file1 ... ]
```

Command Line Options

The options for strings are:

`-t <n>`: specifies the minimum length that a terminated string needs to be before it is printed (default 4).

-n <n>: specifies the minimum length that a non-terminated string needs to be before it is printed (default 8).

-?: Display program description.

Note: If the environment variable STRINGS exists, its value is used to establish default options.

See Also

For related information, see:

About Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the Tail Utility

Use the *Tail* utility to view user-defined number of lines from end of a file.

To Access Tail from the Dashboard

To run Tail from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Tail.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `Tail` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Tail

Tail displays the last <count> lines of each of the files listed on the line. If no files are listed, display the last <count> lines of the standard input.

<count> defaults to 10.

Syntax

```
tail [-<count> | -n <count>] [-fvx?] [ file1 ... ]
```

Command Line Options

The options for Tail are:

-<count> or -n <count>: specifies the number of lines to display.

-f: Follow along forever, outputting appended data as the file grows.

-v: Print headers giving file names.

-x: Expand tabs into the number of spaces specified by the TABS environment variable or into 8 spaces if TABS is undefined.

-?: Display program description.

Note: If the environment variable TAIL exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the -v option, specify -v-.

Note: If the environment variable TABS exists, its value will determine the number of spaces used to expand tabs. Otherwise, tabs are expanded to 8 spaces.

See Also

For related information, see:

About Common Utilities

How to Use the Touch Utility

Use the *Touch* utility to change the creation date/time for any file.

To Access Touch from the Dashboard

To run Touch from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Touch.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the Touch command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Touch

Touch displays the last <count> lines of each of the files listed on the line. If no files are listed, display the last <count> lines of the standard input.

<count> defaults to 10.

Syntax

```
touch [ -c? ] [ mmddhhmm[yy] ] file1 ...
```

Command Line Options

The options for Touch are:

-c: Prevents touch from creating files that did not previously exist

If an argument in the form `mmddhhmm[yy]` is passed to `touch`, the last-write time of each file listed on the command line is set to the specified date and time. If the date and time are not given on the command line, the last-write times of the arguments are set to the current system time. The argument `mmddhhmm[yy]` is formed as follows:

- The first `mm` is a two digit number (between 01 and 12) for the month.
- The `dd` is a two digit number (between 01 and 31) for the day of the month.
- The `hh` is a two digit number (between 00 and 23) for the hour. Note that a 24 hour clock is assumed.
- The second `mm` is a two digit number (between 00 and 59) for the minute.
- The `yy`, if present, is for setting the year. Years between 80 and 99 are understood to be between 1980 and 1999. Years between 00 and 79 are assumed to be for the years 2000 to 2079.

`-?`: Display program description.

Note: If the environment variable `TOUCH` exists, its value is used to establish default options.

You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the `-v` option, specify `-v-`.

See Also

For related information, see:

About Common Utilities

How to Use the Tracert Utility

Tracert (Trace Route) is a route-tracing utility that is used to determine the path that an IP datagram takes to reach a destination. The `tracert` command uses the IP Time-to-Live (TTL) field and ICMP error messages to determine the route from one host to another through a network.

To Access Tracert from the Dashboard

To run Tracert from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Tracert.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `tracert` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Tracert

Syntax

```
tracert ip_address or hostname
```

See Also

For related information, see:

About Common Utilities

Cisco TAC ICM Technical Notes on Cisco.com

How to Use the WC Utility

Use the *WC* utility to view a count of characters, words, or lines in a file.

To Access WC from the Dashboard

To run WC from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > WC.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `wc` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using WC

Syntax

```
wc [ -lwc? ] [ file1 ... ]
```

Command Line Options

The default is for `wc` to count lines, words, and characters for each file given. Counting can be restricted to one or more of lines, words, or characters by using the command line options below:

- l: Enable counting for lines.
- w: Enable counting for words.

-c: Enable counting for characters.

If no files are given, standard input is read for characters.

Example:

`wc file1` counts the lines, words, and characters in file1.

`wc -w -c` counts the words and characters from the standard input.

-?: Display program description.

Note: If the environment variable `WC` exists, its value is used to establish default options.

See Also

For related information, see:

About Common Utilities

How to Use the Which Utility

Use the *Which* utility to view the name of the first file encountered while traversing a path or directory tree.

To Access Which from the Dashboard

To run Which from the Support Tools Dashboard:

1. Use the System Select screen to select the system you want to work with.
2. From the Dashboard menu, select Cisco Common Tools > Which.
3. In the Arguments field, enter desired arguments, as described in the Using section below.
Note: When entering arguments, the `which` command is already implied. Do not enter it in the Arguments field.
4. Specify the command duration time or accept the default (60 seconds). This is the amount of time the command will attempt to run before terminating.
5. If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.
6. Click the Run button.

Using Which

Syntax

```
which [ -a? ] [ -p <environment variable> ] file1 ...
```

Command Line Options

Which will print the full pathname of the first occurrence of a file found in the current directory or along a given path. The default path is the value of the environment variable PATH. If the filename does not have an extension, which will print the first occurrence of `<file>.com`, `<file>.exe`, `<file>.bat`, or `<file>.cmd`, in that order (unless the environment variable PATHEXT is set, in which case its list of extensions is used.)

-a: Prints the full pathnames of all of the files found along the specified path. Note that if the path specifies the same directory more than once (which may include the implicit current directory), the same pathname may occur more than once.

-p: Use the next argument as the name of the environment variable whose value contains the path to search and will not automatically add the extension as described above.

-?: Display program description.

Example

`which -a which` prints the pathnames of all of the places where `which.com`, `which.exe`, `which.bat`, or `which.cmd` exist along the path specified by the value of the environment variable `PATH`.

Note: If the environment variable `WHICH` exists, its value is used to establish default options.

Note: You can override an option that was specified in the environment variable by following the option with a minus '-' sign. For example, to turn off the `-a` option, specify `-a-`.

See Also

For related information, see:

About Common Utilities

How to Use the WinMSD Utility

Use the *WinMSD* utility to view and generate reports that capture detailed Windows NT system configuration information.

Accessing WinMSD

WinMSD is a Microsoft external GUI application present on all Windows 2000 installations. It cannot be run from within the Support Tools dashboard.

To access WinMSD:

1. On the Windows Start bar, click Start > Run.
2. In the Open field, enter `winmsd`.
3. Click OK.

See Also

For related information, see:

[About Common Utilities](#)

[Cisco TAC ICM Technical Notes on Cisco.com](#)

[Microsoft Knowledge Base Search](#)

[How to Create a WinMSD Report \(Microsoft.com\)](#)

[How to Use WINMSDP.EXE \(Microsoft.com\)](#)

Part 5: Reference

How to Stop and Start the Support Tools Dashboard/Tomcat

You stop or restart the Support Tools Dashboard by stopping or restarting the Tomcat Web server on your Support Tools Server. Stopping Tomcat prevents users from accessing the Dashboard.

Support Tools installs Tomcat to run as a service. Thus, the easiest way to stop or restart Tomcat is to stop or restart this service.

To Stop or Start the Support Tools Dashboard/Tomcat

To stop or start the Support Tools Dashboard:

1. On the target machine, open the Windows Services dialog.
2. Select the Cisco CCBU Support Tools HTTP Server service.
3. Right-click on the service and, as appropriate, select Stop or Restart.

See Also

For related information, see:

[How to Stop and Start the Support Tools Server](#)

[How to Stop and Start the Support Tools Node Agent](#)

How to Stop and Start the Support Tools Server

You stop or restart the Support Tools Server by stopping or restarting the Appserver service on your Support Tools Server machine.

When the Appserver service is stopped, users can login to the Support Tools Dashboard but cannot execute any utilities therein. When they attempt to do so, they receive an "Unable to Access Backend Appserver" error.

Note: Under normal conditions it should rarely be necessary to restart the Appserver service: this service starts automatically following installation and upon reboot. It also restarts automatically following a shutdown (for example, due to a crash).

To Stop or Start the Support Tools Server

To stop or start the Support Tools Server:

1. On the target machine, open the Windows Services dialog.
2. Select the Cisco CCBU Support Tools Appserver service.
3. Right-click on the service and, as appropriate, select Stop or Restart.

See Also

For related information, see:

[How to Stop and Start the Support Tools Node Agent](#)

[How to Stop and Start the Support Tools Dashboard/Tomcat](#)

How Stop and Start the Node Agent Service

You stop or restart a Support Tools Node Agent by stopping or restarting the Node Agent service on a Support Tools node. When the Node Agent service is stopped, that machine cannot be managed through Support Tools.

Note: Under normal conditions it should rarely be necessary to restart the Node Agent service: this service starts automatically following installation and upon reboot. It also restarts automatically following a shutdown (for example, due to a crash).

To Stop or Start the Node Agent Service

To stop or start the node agent service on a Support Tools node:

1. On the target machine, open the Windows Services dialog.
2. Select the Cisco CCBU Support Tools Node Agent service.
3. Right-click on the service and, as appropriate, select Stop or Restart.

See Also

For related information, see:

[How to Stop and Start the Support Tools Server](#)

[How to Stop and Start the Support Tools Dashboard/Tomcat](#)

How to Confirm the Support Tools Build Number

As necessary (for example, when making a support call), you can confirm the version and build number of your Support Tools Server from the Support Tools Dashboard.

To Confirm the Support Tools Build Number

To confirm the version and build number of your Support Tools Server:

On the Dashboard menu, select Support Tools. Your version information displays.

See Also

For related information, see:

Cisco Support for Support Tools

Cisco Support for Support Tools

The following resources are available to Support Tools users:

Online Resources

Additional Support Tools information is available online at:

- Latest Support Tools user documentation: www.cisco.com
- Technical tips: www.cisco.com/warp/customer/640/
- Known issues and workarounds: www.cisco.com/cgi-bin/Support/Bugtool/home.pl (listed under Cisco ICM Enterprise Edition)

Note: Some resources on the Cisco Web site require you to have an account. Register for an account at: www.cisco.com/register/

To Open a Technical Assistance Call

You can get technical assistance for Support Tools by contacting Cisco's Technical Assistance Center (TAC).

Providing Information to TAC

To assist you in troubleshooting a problem, the Cisco TAC may ask you to provide the following. You can expedite matters by having it available when you contact them:

1. Your Support Tools version and build number
2. The name of the utility you are having an issue with
3. Details of the behavior you are encountering

To Contact the Cisco TAC

To open a request for technical assistance for Support Tools, contact TAC at:

Online:	www.cisco.com/tac/
---------	--

Email:	tac@cisco.com (please include "Support Tools" in the Subject line)
Telephone:	In North America: 1.800.553.2447 Outside North America: 1.408.526.7209

Copyright

Copyright © 2004, Cisco Systems, Inc. All rights reserved. The Cisco Powered Copyright 2004, CCIP, the Cisco Arrow logo, the Cisco Powered Network mark, the Cisco Systems Verified logo, Cisco Unity, Follow Me Browsing, FormShare, iQ Breakthrough, iQ Expertise, iQ FastTrack, the iQ Logo, iQ Net Readiness Scorecard, Networking Academy, ScriptShare, SMARTnet, TransPath, and Voice LAN are trademarks of Cisco Systems, Inc.; Changing the Way We Work, Live, Play, and Learn, Discover All That's Possible, The Fastest Way to Increase Your Internet Quotient, and iQuick Study are service marks of Cisco Systems, Inc.; and Aironet, ASIST, BPX, Catalyst, CCDA, CCDP, CCIE, CCNA, CCNP, Cisco, the Cisco Certified Internetwork Expert logo, Cisco IOS, the Cisco IOS logo, Cisco Press, Cisco Systems, Cisco Systems Capital, the Cisco Systems logo, Empowering the Internet Generation, Enterprise/Solver, EtherChannel, EtherSwitch, Fast Step, GigaStack, Internet Quotient, IOS, IP/TV, LightStream, MGX, MICA, the Networkers logo, Network Registrar, Packet, PIX, Post-Routing, Pre-Routing, RateMUX, Registrar, SlideCast, StrataView Plus, Stratm, SwitchProbe, TeleRouter, and VCO are registered trademarks of Cisco Systems, Inc. and/or its affiliates in the U.S. and certain other countries.

Tagish JAAS Login Modules 1.0.3

Copyright (C) 2002 –2003 Tagish Ltd. The contents of this file are subject to the Lesser GNU Public License Version 2.1 (the "License"); you may not use this file except in compliance with the License. You may obtain a copy of the License at the following URL: <http://www.gnu.org/copyleft/lesser.html> You can find a copy of the source code for Tagish JAAS Login Modules 1.0.3 at the following URL: <http://free.tagish.net/jaas/> Software distributed under the License is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. See the License for the specific language governing rights and limitations under the License.

Onion Networks Java DIME Library v1.0.3

Copyright (C) 2002 Onion Networks. The contents of this file are subject to the Lesser GNU Public License Version 2.1 (the "License"); you may not use this file except in compliance with the License. You may obtain a copy of the License at the following URL: <http://www.gnu.org/copyleft/lesser.html> You can find a copy of the source code for Java DIME Library v1.0.3 at the following URL: <http://onionnetworks.com/developers/> Software distributed under the License is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. See the License for the specific language governing rights and limitations under the License.

All other trademarks mentioned in this document or Web site are the property of their respective owners. The use of the word partner does not imply a partnership relationship between Cisco and any other company. (0208R)

© 2004 Cisco Systems, Inc.

Doc Version: 1.0103202004

Index

accessing

- Dashboard, 24, 75
- Support Tools, 26
- Support Tools documentation, 6
- utilities, 153

adding

- processes and services, 56
- systems to system list, 73, 78

application server, 119

Appserver service, 68, 278

arp -a utility, 209

AT&T NIC, 153

browsers

- for Dashboard, 27
- using with Dashboard, 73

bugs

- reviewing and submitting, 281

BugTool, 281

build number, 280

CAT utility, 210

CHMOD utility, 212

cicman, 153, 158

Cisco CCBU Support Tools Appserver service, 68

Cisco CCBU Support Tools HTTP Server service, 277

Cisco CCBU Support Tools NodeAgent service, 68

Cisco Common Utilities, 16, 153, 156, 158, 160, 165, 167, 169, 170, 172, 177, 179, 181, 186, 191, 192, 201, 202

Cisco Web utilities, 16, 86, 88, 92, 101, 103, 106, 109, 111, 114, 116, 119, 123, 127, 131, 134, 138, 143, 147

command-line

- using Cisco Web utilities from, 119, 123, 127, 131, 134, 138, 143, 147

command-line mode, 119, 123, 127, 131, 134, 138, 143, 147

common utilities, 16, 209, 210, 212, 214, 216, 218, 221, 223, 225, 228, 231, 233, 235, 236, 239, 241, 243, 244, 246, 248, 250, 251, 253, 255, 257, 259, 261, 262, 263, 264, 266, 268, 270, 271, 273, 275

Compare Registries utility, 86, 96, 119, 138

comparing

- registries, 96, 138

configuring

Dashboard for SSL, 61

distinguished user for Tomcat, 41

ports for Support Tools, 51

processes and services list, 63

SQL for use with System Interrogate utility, 66

- Support Support Tools options**, 56
- Support Tools**, 29, 33, 41
- Support Tools installation options**, 57
- virus scan for Repository**, 60
- connection problems in Dashboard**, 81
- copying**
 - values between registries**, 96, 138
- CP utility**, 214
- Create Log Collections screen**, 114, 147
- Create Log Groups screen**, 109, 143
- creating**
 - log collections**, 114, 147
 - log groups**, 109, 143
 - user groups for Support Tools**, 39
- ctitest**, 153, 160
- customer instances in registry**
 - utilities**, 92, 96, 134, 138
- Dashboard**
 - browser support**, 27
 - configuring for SSL**, 61
 - described**, 12, 24
 - ending sessions**, 80
 - general usage**, 73
 - platforms**, 27
 - privileged utilities**, 16, 24, 26, 39
 - starting and stopping**, 277
 - troubleshooting**, 81

- working with files in**, 94
- Dashboard accessing**, 24, 75
- dbdiff**, 153, 165
- deleting**
 - files from Repository**, 94
 - log collections**, 116, 147
 - deleting log groups**, 111, 143
- DF utility**, 216
- Diff utility**, 218
- distinguished user**, 37, 41
- documentation**
 - for Support Tools**, 6
- downloading**
 - collected logs**, 116, 147
 - files from Repository**, 94
- DU utility**, 221
- dumpcfg**, 153, 167
- editing**
 - log groups**, 111, 143
 - processes and services**, 56
 - processes and services list**, 56
- ending Dashboard sessions**, 80
- errors**
 - encountered in Dashboard**, 81
- features**
 - in Support Tools 1.0**, 10
- FGrep utility**, 223

- files**
 - working with Repository**, 94
- Files screen**, 94
- Findstr utility**, 225
- firewalls**, 14, 37, 51
- Grep utility**, 228
- Head**, 231
- HTTP**, 51
- HTTPS**, 51
- ICM versions**, 27, 29
- icmdba**, 153, 169
- installation**
 - utilities locations**, 16
- installing**
 - Support Tools**, 33, 37, 39, 41, 43, 45, 48, 49, 51, 53
 - Support Tools Node**, 48, 49
 - Support Tools Server**, 43, 45
- instances in registry utilities**, 92, 96, 134, 138
- interactive mode**, 119, 123, 127, 131, 134, 138, 143, 147
- IPConfig -all utility**, 233
- ISQLW utility**, 235
- Java Development Kit**, 27
- JDK**, 27
- local Repository**, 119
- location**
 - of Support Tools installation**, 29
 - utilities**, 16, 86, 119, 153
- log collections**
 - defined**, 106
 - managing**, 116, 147
- Log Collections screen**, 116, 147
- Log Collector**, 86, 106, 109, 111, 114, 116, 119, 143, 147
- log groups**
 - creating**, 109, 143
 - defined**, 106
 - managing**, 111, 143
- Log Groups screen**, 111, 143
- logout of Dashboard**, 80
- logs**
 - collecting with Support Tools**, 106, 109, 111, 114, 116, 143, 147
- LS utility**, 236
- menus**
 - in Dashboard**, 73
- merged logs**
 - defined**, 106
 - network time server requirements**, 37
 - specifying in log collections**, 114, 147
- mptrace**, 153, 170
- MV utility**, 239
- NBTStat utility**, 241
- Net Session utility**, 243
- Net Statistics Server utility**, 244

Net Statistics Workstation utility, 246

NetStat utility, 248

network access to nodes, 43

network time server, 37

network topology, 14

nicroi, 153, 172

nmstart, 153, 177

nmstop, 153, 179

Node Agent

- starting and stopping**, 279

Node Agent Service, 68

nodes

- network access to**, 43

NSLookup utility, 250

NT SS7 NIC PG, 153

NTFS directory privileges, 26, 41, 60

opctest, 153, 181

password

- for Dashboard**, 75

PathPing utility, 251

pcAnywhere, 153

performance, 29, 60

Ping utility, 253

platform support

- for Support Tools 1.0**, 27, 29

ports, 51

privileges

- Dashboard**, 16, 24, 26, 37, 39

Processes and Services, 86, 101, 103, 119, 123, 127

processes and services list, 56, 63

Processes utility, 86, 101, 119, 127

processinfo.xml, 56, 63, 70

procmon, 153, 186

PStat utility, 255

quotation marks, 119

registry, 92, 96, 134, 138

Registry utility, 86, 92, 119, 134, 138

reinstalling Support Tools, 70

renaming

- files in Repository**, 94
- log groups**, 111, 143

Repository

- described**, 12
- local**, 119
- working with files in**, 94

Respository

- excluding from virus scan**, 56, 60

RM utility, 257

Route -print utility, 259

rtrtrace, 153, 191

rttest, 153, 192

saving

- files to Repository**, 94

security, 26, 39, 41

Select System screen, 76

selecting

a target system, 119

an application server, 119

selecting systems, 73, 76, 78, 119

server.xml, 70

services

stopping and starting, 103, 123

Support Tools services, 68, 277, 278, 279

Services utility, 86, 103, 119, 123

Servlet Engine, 27

sessions

ending in Dashboard, 80

Shutdown utility, 261

SQL, 66

SQL 2000, 56

SQLEW utility, 262

ss7nictrace, 153, 201

SSL, 26, 51, 56, 61

starting

Dashboard, 277

Node Agent, 279

services, 103, 123

Support Tools Server, 278

Tomcat, 277

stopping

services, 103, 123

stopping

Dashboard, 277

Node Agent, 279

Support Tools Server, 278

Tomcat, 277

stopping and starting services, 103, 123

Stopshut utility, 263

Strings utility, 264

support for Support Tools, 281

Support Tools

accessing, 26

benefits of, 29

build number, 280

Cisco support for, 281

components of, 12, 29

configuring, 14, 29, 33, 41, 56, 57

Dashboard described, 24

Dashboard using, 24, 73, 75

documentation, 6

features in 1.0, 10

firewalls with, 14, 37, 51

hardware requirements, 43, 48

installation location, 29

installing, 33, 37, 39, 41, 43, 45, 48, 49, 51, 53

network topology, 14

- performance**, 29, 60
- platform support**, 27, 29
- port requirements**, 51
- reinstalling**, 70
- security**, 26, 39, 41
- services**, 68, 277, 278, 279
- software requirements**, 43, 48
- SSL with**, 26, 51, 56, 61
- testing installation**, 53
- uninstalling**, 68
- utilities**, 86, 101, 103, 119, 123, 127, 131, 147, 153
- version**, 280
- viewing and submitting bugs**, 281
- Support Tools Node**
 - described**, 12
 - installing**, 48, 49
 - platforms**, 27
 - uninstalling**, 68
- Support Tools Repository**
 - described**, 12
- Support Tools Server**
 - described**, 12
 - installing**, 43, 45
 - platforms**, 27
 - starting and stopping**, 278
- uninstalling**, 68
- System Interrogate**
 - configuring SQL for use with**, 66
 - using**, 86, 88, 119, 131
- system list**
 - adding systems to**, 76, 78
- System Management screen**, 78
- systemlist.xml**, 70
- systems**
 - adding to system list**, 73, 78
 - selecting for use**, 73, 76, 78, 119
- TAC**, 281
- Tail utility**, 266
- target system**, 119
- telnet**, 86, 119, 153
- terminating processes**, 101, 127
- testing Support Tools installation**, 53
- timeout for Dashboard sessions**, 80
- Tomcat**
 - configuring distinguished user for**, 41
 - installing over previous installation**, 43
 - ports for**, 51
 - starting and stopping**, 277
 - version for Support Tools 1.0**, 27
- topology**
 - Support Tools in network**, 14

- Touch utility**, 268
- Tracert utility**, 270
- troubleshooting**, 81
- uninstalling**
 - Support Tools**, 68
- URL**
 - for Cisco Bug Tool**, 281
 - for Cisco TAC**, 281
 - for Cisco technical documentation**, 6
 - for Dashboard**, 24, 75
- user groups**
 - privileged**, 26, 37, 39
- username**
 - for Dashboard**, 75
- utilities**
 - Cisco common**, 16, 156, 158, 160, 165, 167, 169, 170, 172, 177, 179, 181, 186, 191, 192, 201, 202
 - Cisco Web**, 16, 86, 88, 92, 101, 103, 106, 109, 111, 114, 116, 119, 123, 127, 131, 134, 138, 143, 147
 - common**, 210, 212, 214, 216, 218, 221, 223, 225, 228, 231, 233, 235, 236, 239, 241, 243, 244, 246, 248, 250, 251, 253, 255, 257, 259, 261, 262, 263, 264, 266, 268, 270, 271, 273, 275
 - locations**, 86, 119, 153
 - privileged**, 16
 - troubleshooting**, 81
 - with Support Tools 1.0**, 16
 - version**
 - Support Tools**, 280
 - viewing**
 - files in Repository**, 94
 - log collections**, 116, 147
 - log groups**, 111, 143
 - processes**, 101, 127
 - services**, 103, 123
 - virus scan**, 56, 60
 - vrutrace**, 153, 202
 - WC utility**, 271
 - Web Server**, 27
 - Web site**
 - for Cisco bugs**, 281
 - for Cisco TAC**, 281
 - for Support Tools documentation**, 6
 - web.xml**, 70
 - Which utility**, 273
 - Windows 2000**, 27, 43, 48
 - WinMSD utility**, 275
 - WMI**, 56, 66

Free Manuals Download Website

<http://myh66.com>

<http://usermanuals.us>

<http://www.somanuals.com>

<http://www.4manuals.cc>

<http://www.manual-lib.com>

<http://www.404manual.com>

<http://www.luxmanual.com>

<http://aubethermostatmanual.com>

Golf course search by state

<http://golfingnear.com>

Email search by domain

<http://emailbydomain.com>

Auto manuals search

<http://auto.somanuals.com>

TV manuals search

<http://tv.somanuals.com>