Support Tools Common Tool Reference Guide for ICM/IPCC

Release 2.0(0)

December 2005

Corporate Headquarters
Cisco Systems, Inc.
170 West Tasman Drive
San Jose, CA 95134-1706
USA
http://www.cisco.com
Tel: 408 526-4000
800 553-NETS (6387)
Fax: 408 526-1400

Table of Contents

List of Figures

Preface

Purpose

This document details the use of the Cisco Support Tools Common Tools.

Audience

This document is intended for Users of Cisco Support Tools 2.0(0)

Organization

This document contains three sections:

1. Interactive Mode vs. Batch Mode
2. Cisco Common Tools
3. Using 3rd Party Common Tools

Related Documentation

Additional Documents

- *Support Tools User Guide for Cisco ICM/IPCC*
- *Administration Guide for Cisco IP Contact Center*

Obtaining Documentation

Cisco documentation and additional literature are available on Cisco.com. Cisco also provides several ways to obtain technical assistance and other technical resources. These sections explain how to obtain technical information from Cisco Systems.

Cisco.com

You can access the most current Cisco documentation at this URL:

http://www.cisco.com/techsupport

You can access the Cisco website at this URL:

http://www.cisco.com

You can access international Cisco websites at this URL:

http://www.cisco.com/public/countries_languages.shtml

Product Documentation DVD

Cisco documentation and additional literature are available in the Product Documentation DVD package, which may have shipped with your product. The Product Documentation DVD is updated regularly and may be more current than printed documentation.

The Product Documentation DVD is a comprehensive library of technical product documentation on portable media. The DVD enables you to access multiple versions of hardware and software installation, configuration, and command guides for Cisco products and to view technical documentation in HTML. With the DVD, you have access to the same documentation that is found on the Cisco website without being connected to the Internet. Certain products also have .pdf versions of the documentation available.

The Product Documentation DVD is available as a single unit or as a subscription. Registered Cisco.com users (Cisco direct customers) can order a Product Documentation DVD from the Ordering tool or Cisco Marketplace.

Cisco Ordering Tool:

http://www.cisco.com/en/US/partner/ordering/

Cisco Marketplace:

http://www.cisco.com/go/marketplace/

Ordering Documentation

Beginning June 30, 2005, registered Cisco.com users may order Cisco documentation at the Product Documentation Store in the Cisco Marketplace at this URL::

http://www.cisco.com/go/marketplace/

Cisco will continue to support documentation orders using the Ordering tool:

- Registered Cisco.com users (Cisco direct customers) can order Cisco product documentation from the Ordering tool:

  http://www.cisco.com/en/US/partner/ordering/

- Instructions for ordering documentation using the Ordering tool are at this URL:

  http://www.cisco.com/univercd/cc/td/doc/es_inpck/pdi.htm

- Nonregistered Cisco.com users can order documentation through a local account representative by calling Cisco Systems Corporate Headquarters (California, USA) at 408 526-7208 or, elsewhere in North America, by calling 1 800 553-NETS (6387).

Documentation Feedback

You can rate and provide feedback about Cisco technical documents by completing the online feedback form that appears with the technical documents on Cisco.com.

You can rate and provide feedback about Cisco technical documents by completing the online feedback form that appears with the technical documents on Cisco.com.

You can submit comments by using the response card (if present) behind the front cover of your document or by writing to the following address:

Cisco Systems Attn: Customer Document Ordering 170 West Tasman Drive San Jose, CA 95134-9883

We appreciate your comments.

Cisco Product Security Overview

Cisco provides a free online Security Vulnerability Policy portal at this URL: **http://www.cisco.com/en/US/products/products_security_vulnerability_policy.html**

From this site, you can perform these tasks:

- Report security vulnerabilities in Cisco products.
- Obtain assistance with security incidents that involve Cisco products.
- Register to receive security information from Cisco.

A current list of security advisories and notices for Cisco products is available at this URL: **http://www.cisco.com/go/psirt**

If you prefer to see advisories and notices as they are updated in real time, you can access a Product Security Incident Response Team Really Simple Syndication (PSIRT RSS) feed from this URL: **http://www.cisco.com/en/US/products/products_psirt_rss_feed.html**

Reporting Security Problems in Cisco Products

Cisco is committed to delivering secure products. We test our products internally before we release them, and we strive to correct all vulnerabilities quickly. If you think that you might have identified a vulnerability in a Cisco product, contact PSIRT:

- Emergencies - security-alert@cisco.com

  An emergency is either a condition in which a system is under active attack or a condition for which a severe and urgent security vulnerability should be reported. All other conditions are considered nonemergencies.

- Nonemergencies - psirt@cisco.com

In an emergency, you can also reach PSIRT by telephone:

- 1 877 228-7302
- 1 408 525-6532

Note: We encourage you to use Pretty Good Privacy (PGP) or a compatible product to encrypt any sensitive information that you send to Cisco. PSIRT can work from encrypted information that is compatible with PGP versions 2.x through 8.x. Never use a revoked or an expired encryption key. The correct public key to use in your correspondence with PSIRT is the one that has the most recent creation date in this public key server list: **http://pgp.mit.edu:11371/pks/lookup?search=psirt%40cisco.com&op=index&exact=on**

The link on this page has the current PGP key ID in use.

Obtaining Technical Assistance

Cisco Technical Support provides 24-hour-a-day award-winning technical assistance. The Cisco Technical Support & Documentation website on Cisco.com features extensive online support resources. In addition, if you have a valid Cisco service contract, Cisco Technical Assistance Center (TAC) engineers provide telephone support. If you do not have a valid Cisco service contract, contact your reseller.

Cisco Technical Support & Documentation Website

The Cisco Technical Support & Documentation website provides online documents and tools for troubleshooting and resolving technical issues with Cisco products and technologies. The website is available 24 hours a day, at this URL:

http://www.cisco.com/techsupport

Access to all tools on the Cisco Technical Support & Documentation website requires a Cisco.com user ID and password. If you have a valid service contract but do not have a user ID or password, you can register at this URL:

http://tools.cisco.com/RPF/register/register.do

Note: Use the Cisco Product Identification (CPI) tool to locate your product serial number before submitting a web or phone request for service. You can access the CPI tool from the Cisco Technical Support Website by clicking the **Tools & Resources** Tools. Choose **Cisco Product Identification Tool** from the Alphabetical Index drop-down list, or click the **Cisco Product Identification Tool** RMAs. The CPI tool offers three search options: by product ID or model name; by tree view; or for certain products, by copying and pasting **show** command output. Search results show an illustration of your product with the serial number label location highlighted. Locate the serial number label on your product and record the information before placing a service call.

Submitting a Service Request

Using the online TAC Service Request Tool is the fastest way to open S3 and S4 service requests. (S3 and S4 service requests are those in which your network is minimally impaired or for which you require product information.) After you describe your situation, the TAC Service Request Tool provides recommended solutions. If your issue is not resolved using the recommended resources, your service request is assigned to a Cisco TAC engineer. The TAC Service Request Tool is located at this URL:

http://www.cisco.com/techsupport/servicerequest

For S1 or S2 service requests or if you do not have Internet access, contact the Cisco TAC by telephone. (S1 or S2 service requests are those in which your production network is down or severely degraded.) Cisco TAC engineers are assigned immediately to S1 and S2 service requests to help keep your business operations running smoothly

To open a service request by telephone, use one of the following numbers:

- Asia-Pacific: +61 2 8446 7411 (Australia: 1 800 805 227)
- EMEA: +32 2 704 55 55
- USA: 1 800 553-2447

For a complete list of Cisco TAC contacts, go to this URL:

http://www.cisco.com/techsupport/contacts

Definitions of Service Request Severity

To ensure that all service requests are reported in a standard format, Cisco has established severity definitions.

Severity 1 (S1) -- Your network is down, or there is a critical impact to your business operations. You and Cisco will commit all necessary resources around the clock to resolve the situation.

Severity 2 (S2) -- Operation of an existing network is severely degraded, or significant aspects of your business operation are negatively affected by inadequate performance of Cisco products. You and Cisco will commit full-time resources during normal business hours to resolve the situation.

Severity 3 (S3) -- Operational performance of your network is impaired, but most business operations remain functional. You and Cisco will commit resources during normal business hours to restore service to satisfactory levels.

Severity 4 (S4) -- You require information or assistance with Cisco product capabilities, installation, or configuration. There is little or no effect on your business operations.

Obtaining Additional Publications and Information

Information about Cisco products, technologies, and network solutions is available from various online and printed sources.

- Cisco Marketplace provides a variety of Cisco books, reference guides, and logo merchandise. Visit Cisco Marketplace, the company store, at this URL:

  http://www.cisco.com/go/marketplace/

- Cisco Press publishes a wide range of general networking, training and certification titles. Both new and experienced users will benefit from these publications. For current Cisco Press titles and other information, go to Cisco Press at this URL:

http://www.ciscopress.com

- *Packet* magazine is the Cisco Systems technical user magazine for maximizing Internet and networking investments. Each quarter, Packet delivers coverage of the latest industry trends, technology breakthroughs, and Cisco products and solutions, as well as network deployment and troubleshooting tips, configuration examples, customer case studies, certification and training information, and links to scores of in-depth online resources. You can access Packet magazine at this URL:

  http://www.cisco.com/packet

- *iQ Magazine* is the quarterly publication from Cisco Systems designed to help growing companies learn how they can use technology to increase revenue, streamline their business, and expand services. The publication identifies the challenges facing these companies and the technologies to help solve them, using real-world case studies and business strategies to help readers make sound technology investment decisions. You can access iQ Magazine at this URL:

  http://www.cisco.com/go/iqmagazine

- *Internet Protocol Journal* is a quarterly journal published by Cisco Systems for engineering professionals involved in designing, developing, and operating public and private internets and intranets. You can access the Internet Protocol Journal at this URL:

  http://www.cisco.com/ipj

- World-class networking training is available from Cisco. You can view current offerings at this URL:

  http://www.cisco.com/en/US/learning/index.html

Chapter 1

Interactive Mode vs. Batch Mode

Support Tools 2.0 supports two primary modes of operation; **Interactive-Mode** which is the standard mode to use when querying a single system, and **Batch Mode** which allows you to query several systems at a time for any of the specific tools.

Interactive Mode immediately queries the selected system when you invoke the selected tool.

Batch Mode allows you to schedule jobs for multiple systems at any point in the future.

This section contains the following topics:

Working in Interactive Mode

Using interactive mode simply involves selecting the system that you want to query using the **Select System** tool, then using any of the tools that fall under the **Interactive Mode** menu.

For some tools, such as System Interrogate, more detail is available in Interactive Mode because the support tools server first queries the selected system to determine the additional types of information that can be retrieved and presents these choices to the user. Whereas in batch mode this cannot be done since several systems of different node types may be queried during a batch mode process, and there is no user-intervention to select specific results from a server query.

Working in Batch Mode

Batch Mode allows you to query several servers at once by scheduling a tool to run at any time in the future.

Using Tools in Batch Mode

To use a tool in batch mode:

Step 1 Select a tool to use that is within the **Batch Mode** menu.

Step 2 Schedule a time to run the tool. Enter a Date and Time using the calendar at the top of the page for the selected Batch Mode tool.

Note: You must select a time in the future.

Step 3 Check the select boxes next to the Systems on which you want use this tool.

Step 4 Click **Schedule** to schedule the Batch Mode operation.

A page appears confirming that the batch job has been scheduled.

Step 5 Click **OK**.

You are returned to the Batch Mode scheduling screen for the selected tool.

Note:

- You can view the results of batch mode operations for any given tool by selecting **History** under that tool in the **Batch Mode** node of the Support Tools menu.
- You can view **Pending Jobs** for any given batch Mode tool by selecting the **Pending Jobs** node under that tool in the **Batch Mode** node of the Support Tools menu.
- A separate batch job is created in the **Pending** list for each system that is selected.

Pending Jobs

The **Pending Jobs** screen displays a list of jobs that are schedule to run for the selected Batch Mode tool.

You can click **Refresh** at any time to see if any new jobs have been scheduled or started.

Note: If you schedule a job in the short future (~ 2 minutes) then the job may not appear in the Pending Jobs prior to its execution.

Canceling a Batch Mode Job

You can cancel any pending job for a batch mode tool. To cancel a pending job:

Step 1 Check the select box(es) next to the scheduled time of a pending job.

Step 2 Click **Cancel**.

A confirmation dialog box appears.

Step 3 Click **OK** in the confirmation dialog box.

Step 4 Click **OK** to return to the Pending Jobs screen.

Chapter 2

Cisco Common Tools

Cisco Common Tools are a set of Cisco-authored utilities designed for use with the ICM components supported by Support Tools 2.0.

Most Cisco Common Tools 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 Tools 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

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 Tools 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	
procmon	General-purpose debugging tool. Can be used in conjunction with various Network Interface Controller (NIC) and Peripheral Interface Manager	PGs, Call Routers	X	

Utility Name:	Description:	Installed On:	Dashboard Privileged Users Only:	External GUI--No Dashboard or Command-Line Access:
	(PIM) processes. Use to verify status, set debug trace bits, etc.			
rtrtrace	Use to set various debug tracing on a CallRouter. The additional tracing is output to the .EMS log files, and viewed with the Trace and Log tools..	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		

This section contains the following topics:

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access CICMan from the Dashboard

To run CICMan from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > CICMan.

Step 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.

Step 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.

Step 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.

Step 6 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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, gmeters: 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.

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access CTITest from the Dashboard

To run CTITest from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system (namely, PG) you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > CTITest.

Step 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.

Step 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.

Step 5 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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 - Configuring

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 (icrmsgs.dll) is also located in the <icm_root>\bin directory. Because icrmsgs.dll is an active services file, it is recommended that these two files be copied into separate directory.

Step 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>`**

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

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

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

2. 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.

```
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.
- If it is a simplex environment, use "" to specify a null value.

Step 3 Configure the CTI protocol version to 6: **`c:\abc\>ctitest: config /version 6`**

Step 4 Configure the service mask to 7: **`c:\abc\>ctitest: config /service 7`**

Step 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`**

Step 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.

Using CTITest - 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 CCTime 13:52:32 PGStatus NORMAL Peripheral:
Online
```

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

Using CTITest - 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.

Using CTITest - Logging In

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]`**

Command:	Parameters:
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]**

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access DBDiff from the Dashboard

To run DBDiff from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system (namely, PG) you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > DBDiff.

Step 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.

Step 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.

Step 5 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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>`**

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 geoxyzlgra cust1_sideB geoxyzlgrb`**

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}"]`**

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access DumpCfg from the Dashboard

To run DumpCfg from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system (namely, PG) you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > DumpCfg.

Step 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.

Step 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.

Step 5 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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
```

```
c:\>dumpcfg
```

Using DumpCfg

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

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).

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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`**.

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access MPTrace from the Dashboard

To run MPTrace from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > MPTrace.

Step 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.

Step 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.

Step 5 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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\merpincap> mptrace
```

Using MPTrace - Command Line Options

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.

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] [/?]`**

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access NICROI from the Dashboard

To run NICROI from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > NICROI.

Step 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.

Step 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.

Step 5 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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>xyzzyf 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 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).

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.

Capturing NICROI Data to niclog.xxx

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:

Step 1 Type **`mgmt help`** within the NICROI window to see the options available.

Step 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.

Step 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.

Capturing NICROI Data to Roilog.txt

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 NICROI 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
```

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access NMStart from the Dashboard

To run NMStart from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > NMStart.

Step 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.

Step 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.

Step 5 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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`**

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:

- This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

- 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:

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > NMStop.

Step 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.

Step 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.

Step 5 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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`**

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access OPCTest from the Dashboard

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

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > OPCTest.

Step 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.

Step 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.

Step 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.

Step 6 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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.

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.

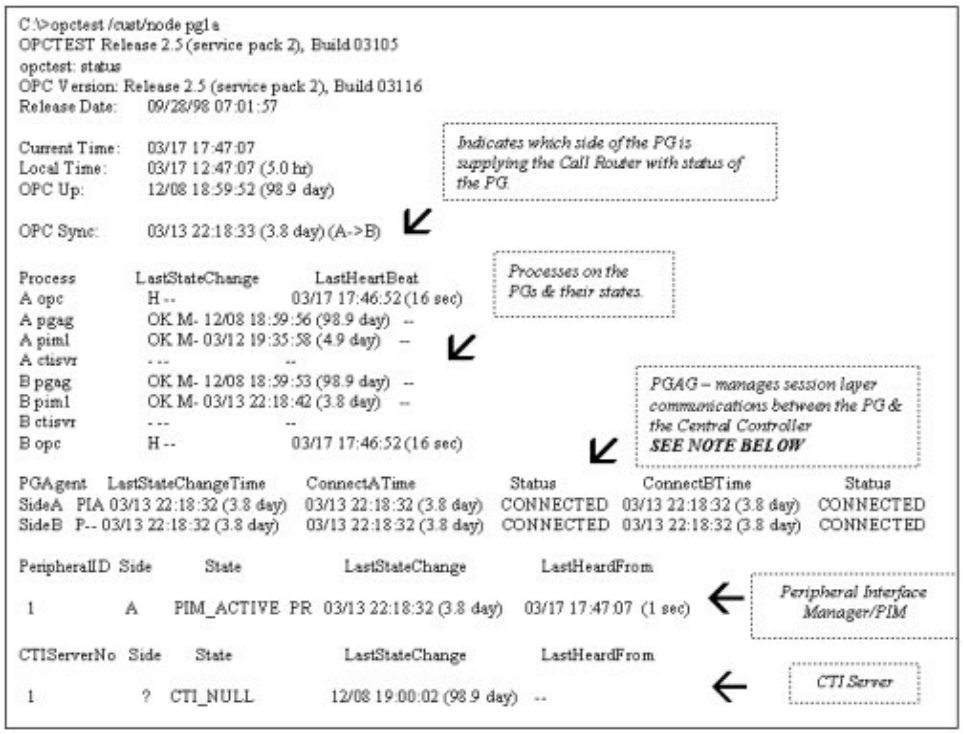
trunk_group_trace: Controls TrunkGroup tracing in the OPC.

tuning_param, tune: Controls tuning in the router.

Example

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

Figure 1: OPC Test Example Output



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 Trace and Log utility to view the output of the EMS logs.

For example:

```
opctest: debug /? Usage: debug_control [/realtime] [/agent] [/halfhour]
[/rcmeter] [/routing] [/skillgroup] [/closedcalls] [/cstaecr] [/cstacer]
[/pimmsg] [/ctimsg] [/rcmsg] [/dmpmsg] [/icmsg] [/opcmsg] [/mdsmsg]
[/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.

Warning: 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.

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access Procmon from the Dashboard

To run Procmon from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > Procmon.

Step 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.

Step 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.

Step 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.

Step 6 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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] [/?]`**

Procmon 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

Procmon 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 **/?**.

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

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 Trace and Log 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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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`**.

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

To Access RTTest from the Dashboard

To run RTTest from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > RTTest.

Step 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.

Step 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.

Step 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.

Step 6 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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 csco /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: 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 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 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 03/06 11:10:20 (11.2 day)	
B agi	- --	

Process:	LastStateChange:	LastHeartBeat:
B cic	- --	
B csfs	OK 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 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 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.

configgnic: 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 Trace and Log 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 registry settings. Make sure there is enough disk space available before increasing log file capacities.

  To enter the Windows registry:

  a. From a command prompt, enter regedt32.

  b. 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\csco\routera\ems\currentversion\library\processes\rtr\EMSAllLogFilesMax

  \\.\software\geotel\icr\csco\routera\ems\currentversion\library\processes\rtr\EMSLogFileMax

  The first parameter - **EMSAllLogFilesMax** 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 **EMSAllLogFilesMax** 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`**

How to Use the SS7NICTrace Utility

Use SS7NICTrace to view and set debug trace bits on an ICM SS7 NIC PG.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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`**.

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.

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

For use with ICM VRU PGs only.

To Access VRUTrace from the Dashboard

To run VRUTrace from the Support Tools Dashboard:

Step 1 Use the System Select screen to select the system you want to work with.

Step 2 From the Dashboard menu, select Cisco Common Tools > VRUTrace.

Step 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.

Step 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.

Step 5 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 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] [/?]**

VRUTrace 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
```

Chapter 3

Using 3rd Party Common Tools

3rd Party Common Tools 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.

These GUI-based third-party tools include:

- isql/w
- sqlew
- winmsd

Almost all 3rd Party Common Tools are installed on all Support Tools nodes/Windows OSs. Several (see the table in the next section) may be particular to SQL Server installations.

Most 3rd Party Common Tools 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 3rd Party Common Tools 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).

3rd Party Common Tools At a Glance

Utility Name:	Description:	Supported Node Operating Systems:
Arp-a	Use to find the media access control address of a host on the same physical network.	Windows 2000, Windows 2003, Windows XP

Utility Name:	Description:	Supported Node Operating Systems:
cat	Use the CAT utility to display, print, and combine files. Equivalent of the DOS type command.	Windows 2000, Windows 2003, Windows XP
chmod	Use to change file permissions. Equivalent of the DOS attrib command.	Windows 2000, Windows 2003, Windows XP
cp	Equivalent of the DOS copy command.	Windows 2000, Windows 2003, Windows XP
df	Use to view disk free information.	Windows 2000, Windows 2003, Windows XP
diff	Use to compare two text files or two directories.	Windows 2000, Windows 2003, Windows XP
du	Use to view disk usage information.	Windows 2000, Windows 2003, Windows XP
fgrep	Use to search for text patterns within a set of files.	Windows 2000, Windows 2003, Windows XP
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.	Windows 2000, Windows 2003, Windows XP
grep	Use to search for text patterns within a set of files.	Windows 2000, Windows 2003, Windows XP
head	Use to view user-defined number of lines from the start of a file.	Windows 2000, Windows 2003, Windows XP
IPConfig -all	Use to get host computer configuration information, including the IP address, subnet mask, and default gateway.	Windows 2000, Windows 2003, Windows XP
isql/w	Interactive SQL for Windows. Use to query and update Microsoft SQL Server databases.	Windows 2000, Windows 2003, Windows XP
ls	Use to view directory listings. Equivalent of the DOS dir command.	Windows 2000, Windows 2003, Windows XP
mv	Use to move files and directories. Equivalent of the DOS move or rename command.	Windows 2000, Windows 2003, Windows XP
NBTStat	Use to troubleshoot NetBIOS name resolution problems.	Windows 2000, Windows 2003, Windows XP
Net Session	Use to view information about all current client sessions with the current system.	Windows 2000, Windows 2003, Windows XP

Utility Name:	Description:	Supported Node Operating Systems:
NetStat	Use to view network information for the current system, including protocol statistics and current TCP/IP network connections.	Windows 2000, Windows 2003, Windows XP
Net Statistics Server	Use to view statistics for the local Server service.	Windows 2000, Windows 2003, Windows XP
Net Statistics Works	Use to view statistics for the local Workstation service.	Windows 2000, Windows 2003, Windows XP
NSLookup	Use to get the IP address and fully qualified DNS of the current system's DNS name server.	Windows 2000, Windows 2003, Windows XP
PathPing	A route tracing tool that combines features of the ping and tracert commands with additional information.	Windows 2000, Windows 2003, Windows XP
ping	Low-level connectivity test. Use to verify end-to-end TCP/IP network infrastructure.	Windows 2000, Windows 2003, Windows XP
pstat	Use to list all NT or Win2K processes, their process ID (PID), memory and cpu utilization, etc.	Windows 2000, Windows 2003, Windows XP
rm	Use to delete files and directories. Equivalent of the DOS delete, recursive del subdirectories commands.	Windows 2000, Windows 2003, Windows XP
Route-print	Use to view the contents of the network routing tables.	Windows 2000, Windows 2003, Windows XP
shutdown	Use to shut down a remote host. When used from the Dashboard, shutdown will automatically restart the host after 60 seconds.	Windows 2000, Windows 2003, Windows XP
sqlew	SQL Enterprise Manager. Use to monitor and update SQL Server operating parameters, expand databases, etc.	Windows 2000, Windows 2003, Windows XP
stopshut	Use to stop a system from shutting down due to an application or operating system failure.	Windows 2000, Windows 2003, Windows XP
strings	Use to find printable strings in a binary file.	Windows 2000, Windows 2003, Windows XP
tail	Use to view user-defined number of lines from end of a file.	Windows 2000, Windows 2003, Windows XP
touch	Use to change the creation date/time for any file.	Windows 2000, Windows 2003, Windows XP

Utility Name:	Description:	Supported Node Operating Systems:
tracert	Related to ping command. Use to show each intermediary Hop in the TCP/IP network. Useful in troubleshooting LAN and WAN faults.	Windows 2000, Windows 2003, Windows XP
wc	Use to view a count of characters, words, or lines in a file.	Windows 2000, Windows 2003, Windows XP
which	Use to view the name of the first file encountered while traversing a path or directory tree.	Windows 2000, Windows 2003, Windows XP
winmsd	Windows Diagnostics. Use to gather detailed configuration information about computers running Windows.	Windows 2000, Windows 2003, Windows XP
NetshDump	Allows you to run the NetShell utility to display or modify the configuration of a currently running computer.	Windows 2000, Windows 2003, Windows XP
Bootcfgqry	Queries Boot.ini file settings.	Windows 2003, Windows XP
Defraganalyze	Analyzes boot files, data files, and folders on local volumes for fragmentation.	Windows 2003, Windows XP
Defragreport	Reports fragmentation of boot files, data files, and folders on local volumes.	Windows 2003, Windows XP
Driverquery	Displays a list of all installed device drivers and their properties.	Windows 2003, Windows XP
Eventtriggers	Displays and configures event triggers on local or remote machines.	Windows 2003, Windows XP
Getmac	Returns the media access control (MAC) address and list of network protocols associated with each address for all network cards in each computer, either locally or across a network.	Windows 2003, Windows XP
Logman	Manages and schedules performance counter and event trace log collections on local and remote systems.	Windows 2003, Windows XP
Openfiles	Queries or displays open files. Also queries, displays, or disconnects files opened by network users.	Windows 2003, Windows XP
Relog	Extracts performance counters from performance counter logs into other formats, such as text-TSV (for tab-delimited text), text-CSV (for comma-delimited text), binary-BIN, or SQL.	Windows 2003, Windows XP
SC	Communicates with the Service Controller and installed services. SC.exe retrieves and sets control information about services.	Windows 2003, Windows XP
Schtasks	Schedules commands and programs to run periodically or at a specific time. Adds and removes tasks from the schedule, starts and stops tasks on demand, and displays and changes scheduled tasks.	Windows 2003, Windows XP

Utility Name:	Description:	Supported Node Operating Systems:
SysteminfoTable	Displays detailed configuration information about a computer and its operating system, including operating system configuration, security information, product ID, and hardware properties, such as RAM, disk space, and network cards.	Windows 2003, Windows XP
SysteminfoList	Displays detailed configuration information about a computer and its operating system, including operating system configuration, security information, product ID, and hardware properties, such as RAM, disk space, and network cards.	Windows 2003, Windows XP
SysteminfoCSV	Displays detailed configuration information about a computer and its operating system, including operating system configuration, security information, product ID, and hardware properties, such as RAM, disk space, and network cards.	Windows 2003, Windows XP
Taskkill	Ends one or more tasks or processes. Processes can be killed by process ID or image name.	Windows 2003, Windows XP
TasklistTable	Displays a list of applications and services with their Process ID (PID) for all tasks running on either a local or a remote computer.	Windows 2003, Windows XP
Tasklist	Displays a list of applications and services with their Process ID (PID) for all tasks running on either a local or a remote computer.	Windows 2003, Windows XP
TasklistCSV	Displays a list of applications and services with their Process ID (PID) for all tasks running on either a local or a remote computer.	Windows 2003, Windows XP

This section contains the following topics:

Arp -a

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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 IPÃƒâ€šÃ‚toÃƒâ€šÃ‚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.

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.

Options for this tool:

1. **Command duration (seconds)** - Enter the max execution time for this command.
2. **Elevate command priority on remote system** - Cause command to run at a higher priority.

cat

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the CAT utility to display, print, and combine files.

To Access CAT from the Dashboard

To run CAT from the Support Tools Dashboard:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using CAT - Command Line Options

```
cat [ -usx? ] [ -v [-t] [-e] ] file1 ...
```

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 ^<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 '> null', 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.

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-.

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.

chmod

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the CHMOD utility to set file permissions.

To Access CHMOD from the Dashboard

To run CHMOD from the Support Tools Dashboard:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

When entering arguments, the CHMOD command is already implied. Do not enter it in the Arguments field.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using CHMOD - Command Line Options

```
chmod <mode> file1 ...
```

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.

cp

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the CP utility to copy files.

To Access CP from the Dashboard

To run CP from the Support Tools Dashboard:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using CP - Command Line Options

```
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.

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 if 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.
- 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-.
- The order of options f and I is significant: the one specified last determines whether a confirmation is requested.

df

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using DF - Command Line Options

```
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:.

The options for DF are:

-a: Print free space for all loadable drives.

-?: Display program description.

Note:

- If the environment variable DF 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 -a option, specify -a-.
- Option -a can be specified twice to restore normal operation, namely, to undo the effect of specifying option -a in the environment variable.

diff

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Diff - Command Line Options

```
diff [ -rebidtvo? ] [-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.

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 0fbe' 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.
- 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-.

du

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using DU - Command Line Options

```
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)

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.
- 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-.

fgrep

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using FGrep - Command Line Options

```
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)

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.

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-.

FindStr

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Findstr - Command Line Options

```
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)

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

grep

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Grep - Command Line Options

```
grep [ -mvclinfse? ] <pattern> file1 ...
```

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.
- 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-.

head

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Head - Command Line Options

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.

```
head [-<count> | -n <count>] [-vx?] [ file1 ... ]
```

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.
- 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-.
- 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.

IPConfig /all

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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.

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.

ISQL

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

ISQL is an application used to query and update Microsoft SQL Server databases.

To Access ISQL from the Dashboard

To run ISQL from the Support Tools Dashboard:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using ISQL

ISQL uses the following options:

- **[-U login id]**
- **[-P password]**
- **[-S server]**
- **[-H hostname]**
- **[-E trusted connection]**
- **[-d use database name]**
- **[-l login timeout]**
- **[-t query timeout]**
- **[-h headers]**
- **[-s colseparator]**
- **[-w columnwidth]**
- **[-a packetsize]**
- **[-e echo input]**
- **[-x max text size]**
- **[-L list servers]**
- **[-c cmdend]**
- **[-q "cmdline query"]**
- **[-Q "cmdline query" and exit]**
- **[-n remove numbering]**
- **[-m errorlevel]**
- **[-r msgs to stderr]**
- **[-i inputfile]**
- **[-o outputfile]**
- **[-p print statistics]**

- **[-b On error batch abort]**
- **[-O use Old ISQL behavior disables the following] <EOF> batch processing Auto console width scaling Wide messages default errorlevel is -1 vs 1**
- **[-? show syntax summary (this screen)]**

ISQLW

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 On the Windows Start bar, click Start > Run.

Step 2 In the Open field, enter **isqlw**.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click OK.

ls

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the LS utility to view directory listings.

To Access LS from the Dashboard

To run LS from the Support Tools Dashboard:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using LS - Command Line Options

```
ls [ -cx1lmfetsurRpadoM? ] [ file1 ... ]
```

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.
- 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-.

mv

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the MV utility to move files and directories.

To Access MV from the Dashboard

To run MV from the Support Tools Dashboard:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using MV - Command Line Options

```
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.

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.
- 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-.
- The order of options f and i is significant: the one specified last determines whether a confirmation is requested.
- If the environment variable MV exists, its value is used to establish default options.

NBTStat

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using NBTStat - Command Line Options

```
nbtstat [ [-a RemoteName] [-A IP address] [-c] [-n] [-r] [-R] [-RR]
[-s] [-S] [interval] ]
```

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.

Net Session

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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.

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.

Net Statistics Server

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the Net Statistics Works utility to view statistics for the local Server service.

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.

Net Statistics Workstation

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the Net Statistics Workstation utility to view statistics for the local Workstation service.

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.

NetStat

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using NetStat - Command Line Options

```
NetStat [-a] [-e] [-n] [-s] [-p proto] [-r] [interval]
```

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.

NSLookUp

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using NSLookup - Command Line Options

```
nslookup ip_address or hostname
```

Example: **`nslookup www.cisco.com`**

PathPing

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using PathPing - Command Line Options

```
pathping [-n] [-h maximum_hops] [-g host-list] [-p period] [-q
num_queries] [-w timeout] [-t] [-R] [-r] target_name
```

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.

Ping

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Ping - Command Line Options

```
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
```

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.

PStat

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the PStat utility to list all the processes running on a Microsoft Windows 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.
- 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.

rm

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the RM utility to delete files and directories.

To Access RM from the Dashboard

To run RM from the Support Tools Dashboard:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using RM - Command Line Options

```
rm [ -firv? ] file ...
```

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.
- 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-.

Route -PRINT

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the Route -print utility to view the contents of the network routing tables.

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.

Shutdown Tool

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the Shutdown Tool utility to stop and restart a remote host after a 60 second grace period.

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.

StopShut

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Stopshut - Command Line Options

```
c:\>stopshut
```

SQLEW

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 On the Windows Start bar, click Start > Run.

Step 2 In the Open field, enter sqlew.

Step 3 Click OK.

Strings

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Strings - Command Line Options

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.

```
strings [ -? ] [ -t <n>] [ -n <n>] [ file1 ... ]
```

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.

tail

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Tail - Command Line Options

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.

```
tail [-<count> | -n <count>] [-fvx?] [ file1 ... ]
```

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.
- 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-.
- 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.

touch

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Touch - Command Line Options

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.

```
touch [ -c? ] [ mmddhhmm[yy] ] file1 ...
```

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-.

Tracert

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Tracert - Command Line Options

```
tracert ip_address or hostname
```

Example: `tracert www.cisco.com`

WC

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using WC - Command Line Options

```
wc [ -lwc? ] [ file1 ... ]
```

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.

-?: Display program description.

Note: If the environment variable WC exists, its value is used to establish default options.

which

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

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:

Step 1 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.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Which - Command Line Options

```
which [ -a? ] [ -p <environment variable> ] file1 ...
```

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.

Note:

- If the environment variable WHICH 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 -a option, specify -a-.

Winmsd

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Use the WinMSD utility to view and generate reports that capture detailed Windows 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.

Step 1 On the Windows Start bar, click Start > Run.

Step 2 In the Open field, enter **`winmsd`**.

Step 3 Click OK

NetshDump

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Netsh is a command-line scripting utility that allows you to, either locally or remotely, display the network configuration of a computer that is currently running.

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.

Bootcfgqry

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays the Boot.ini file settings of the selected system..

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.

Defraganalyze

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Analyzes the volumes (hard drives) on the selected system and determines whether or not a defrag is recommended for any system volumes.

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.

Defragreport

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays a volume (hard drive) fragmentation report for the hard drives on the selected system.

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.

Driverquery

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays a list of all installed device drivers and their properties.

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.

Eventtriggers

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

This tool enables an administrator to display and configure "Event Triggers" on local or remote systems.

To Access Eventtriggers from the Dashboard

To run Eventtriggers from the Support Tools Dashboard:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Note: When entering arguments, the Eventtriggers command is already implied. Do not enter it in the Arguments field.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Eventtriggers - Command Line Options

```
EVENTTRIGGERS /parameter [arguments]
```

The options for Eventtriggers are:

/Create: Create a new Event Trigger.

/Delete: Deletes an Event Trigger by its trigger ID.

/Query: Displays the Event Trigger properties and settings.

/?: Displays help/usage.

Examples (Addition Command Line Option help is provided in the tool by using the following examples):

```
EVENTTRIGGERS /Create /?

EVENTTRIGGERS /Delete /?

EVENTTRIGGERS /Query /?
```

Getmac

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Returns the media access control (MAC) address and list of network protocols associated with each address for all network cards in each computer, either locally or across a network.

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.

Logman

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Manages and schedules performance counter and event trace log collections on local and remote systems.

To Access Logman from the Dashboard

To run Logman from the Support Tools Dashboard:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Note: When entering arguments, the logman command is already implied. Do not enter it in the Arguments field.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Logman - Command Line options

```
logman VERB <collection_name>
```

The command line options forLogman are:

Verbs:

- create <counter|trace>: Create a new collection.
- start: Start an existing collection and set the begin time to manual.
- stop: Stop an existing collection and set the end time to manual.
- delete: Delete an existing collection.
- query [collection_name|providers]: Query collection properties. If no collection_name is given all collections are listed. The keyword 'providers' will list all of the registered Event Trace providers.
- update: Update an existing collection properties.

Parameters:

- <collection_name>: Name of the collection.

Options:

- -? Displays context sensitive help.
- -s <computer> Perform the command on specified remote system.
- -config <filename> Settings file containing command options.
- -b <M/d/yyyy h:mm:ss[AM|PM]> Begin the collection at specified time.
- -e <M/d/yyyy h:mm:ss[AM|PM]> End the collection at specified time.
- -m <[start] [stop]> Change to manual start or stop rather than a scheduled begin or end time.
- -[-]r Repeat the collection daily at the specified begin and end times.
- -o <path|dsn!log> Path of the output log file or the DSN and log set name in a SQL database.
- -f <bin|bincirc|csv|tsv|sql> Specifies the log format for the collection.
- -[-]a Append to an existing log file.
- -[-]v [nnnnnn|mmddhhmm] Attach file versioning information to the end of the log name.
- -[-]rc <filename> Run the command specified each time the log is closed.
- -[-]max <value> Maximum log file size in MB or number of records for SQL logs.
- -[-]cnf [[[hh:]mm:]ss] Create a new file when the specified time has elapsed or when the max size is exceeded. -c <path [path ...]> Performance counters to collect.
- -c <path [path ...]> Performance counters to collect.
- -cf <filename> File listing performance counters to collect, one per line.
- -si <[[hh:]mm:]ss> Sample interval for performance counter collections.
- -ln <logger_name> Logger name for Event Trace Sessions.
- -[-]rt Run the Event Trace Session in real-time mode.
- -p <provider [flags [level]]> A single Event Trace provider to enable.
- -pf <filename> File listing multiple Event Trace providers to enable.
- -[-]ul Run the Event Trace Session in user mode.
- -bs <value> Event Trace Session buffer size in kb.
- -ft <[[hh:]mm:]ss> Event Trace Session flush timer.
- -nb <min max> Number of Event Trace Session buffers.

- -fd Flushes all the active buffers of an existing Event Trace Session to disk.
- -[-]u [user [password]] User to Run As. Entering a * for the password produces a prompt for the password. The password is not displayed when you type it at the password prompt.
- -rf <[[hh:]mm:]ss> Run the collection for specified period of time.
- -y Answer yes to all questions without prompting.
- -ets Send commands to Event Trace Sessions directly without saving or scheduling.
- -mode <trace_mode [trace_mode ...]> Event Trace Session logger mode.
- -ct <perf|system|cycle> Event Trace Session clock type.

Note: Where [-] is listed, an extra - negates the option. For example --r turns off the -r option.

Examples:

```
logman create counter perf_log -c "\Processor(_Total)\% Processor Time"

logman create trace trace_log -nb 16 256 -bs 64 -o c:\logfile

logman start perf_log

logman update perf_log -si 10 -f csv -v mmddhhmm

logman update trace_log -p "Windows Kernel Trace" (disk,net)
```

Openfiles

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Queries or displays open files. Also queries, displays, or disconnects files opened by network users.

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.

Relog

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Extracts performance counters from performance counter logs into other formats, such as text-TSV (for tab-delimited text), text-CSV (for comma-delimited text), binary-BIN, or SQL.

Relog creates new performance logs from data in existing performance logs by changing the sampling rate and/or converting the file format. Supports all performance log formats, including Windows compressed logs.

To Access Relog from the Dashboard

To run Relog from the Support Tools Dashboard:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Note: When entering arguments, the relog command is already implied. Do not enter it in the Arguments field.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Relog - Command Line Options

```
relog <filename [filename ...]>
```

Options for relog:

-? Displays context sensitive help.

-a Append output to existing binary file.

-c <path [path ...]> Counters to filter from input log.

-cf <filename> File listing performance counters to filter form the input log. Default is all counters in the original log file.

-f <CSV|TSV|BIN|SQL> Output file format.

-t <value> Only write every nth record into the output file. Default is to write every record.

-o Output file path or SQL database.

-b <M/d/yyyy h:mm:ss[AM|PM]> Begin time for the first record to write into the output file.

-e <M/d/yyyy h:mm:ss[AM|PM]> End time for the last record to write into the output file.

-config <filename> Settings file containing command options.

-q List performance counters in the input file.

-y Answer yes to all questions without prompting.

Examples:

```
relog logfile.csv -c "\Processor(_Total)\% Processor Time" -o
logfile.blg

relog logfile.blg -cf counters.txt -f bin

relog logfile.blg -f csv -o logfile.csv -t 2

relog logfile.blg -q -o counters.txt
```

SC

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Communicates with the Service Controller and installed services. SC.exe retrieves and sets control information about services.

```
sc <server> [command] [service name] <option1> <option2>...
```

The option <server> has the form "**\\ServerName**".

Commands for SC:

query: Queries the status for a service, or enumerates the status for types of services.

queryex: Queries the extended status for a service, or enumerates the status for types of services.

start: Starts a service.

pause: Sends a PAUSE control request to a service.

interrogate: Sends an INTERROGATE control request to a service.

continue: Sends a CONTINUE control request to a service.

stop: Sends a STOP request to a service.

config: Changes the configuration of a service (persistent).

description: Changes the description of a service.

failure: Changes the actions taken by a service upon failure.

qc: Queries the configuration information for a service.

qdescription: Queries the description for a service.

qfailure: Queries the actions taken by a service upon failure.

delete: Deletes a service (from the registry).

create: Creates a service. (adds it to the registry).

control: Sends a control to a service.

sdshow: Displays a service's security descriptor.

sdset: Sets a service's security descriptor.

GetDisplayName: Gets the DisplayName for a service.

GetKeyName: Gets the ServiceKeyName for a service.

EnumDepend: Enumerates Service Dependencies.

The following commands don't require a service name:

Usage: **`sc <server> <command> <option>`**

boot: (ok | bad) Indicates whether the last boot should be saved as the last-known-good boot configuration

Lock: Locks the Service Database

QueryLock: Queries the LockStatus for the SCManager

Schtasks

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Schedules commands and programs to run periodically or at a specific time. Adds and removes tasks from the schedule, starts and stops tasks on demand, and displays and changes scheduled tasks.

To Access Schtasks from the Dashboard

To run Relog from the Support Tools Dashboard:

Step 1 In the Arguments field, enter desired arguments, as described in the Using section below.

Note: When entering arguments, the Schtasks command is already implied. Do not enter it in the Arguments field.

Step 2 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.

Step 3 If desired, check Elevate Command Priority. This ensures the command will run regardless of the level of server activity.

Step 4 Click the Run button.

Using Schtasks - Command Line Options

Options for Schtasks are:

/Create Creates a new scheduled task.

/Delete Deletes the scheduled task(s).

/Query Displays all scheduled tasks.

/Change Changes the properties of scheduled task.

/Run Runs the scheduled task immediately.

/End Stops the currently running scheduled task.

/? Displays help message.

Examples:

Note: Run the commands with the following options to receive additional help.

```
SCHTASKS

SCHTASKS /?

SCHTASKS /Run /?

SCHTASKS /Create /?

SCHTASKS /Delete /?

SCHTASKS /Query /?

SCHTASKS /Change /?
```

SysteminfoTable

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays detailed configuration information about a computer and its operating system, including operating system configuration, security information, product ID, and hardware properties, such as RAM, disk space, and network cards.

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.

SysteminfoList

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays detailed configuration information about a computer and its operating system, including operating system configuration, security information, product ID, and hardware properties, such as RAM, disk space, and network cards.

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.

SysteminfoCSV

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays detailed configuration information about a computer and its operating system, including operating system configuration, security information, product ID, and hardware properties, such as RAM, disk space, and network cards.

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.

Taskkill

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Ends one or more tasks or processes. Processes can be killed by process ID or image name.

Usage: **`TASKKILL [/S system [/U username [/P [password]]]] { [/FI filter] [/PID processid | /IM imagename] } [/T] [/F]`**

Parameter List:

/S system: Specifies the remote system to connect to.

/U [domain\]user: Specifies the user context under which the command should execute.

/P [password]: Specifies the password for the given user context. Prompts for input if omitted.

/FI filter: Applies a filter to select a set of tasks. Allows "*" to be used. ex. imagename eq acme*

/PID processid: Specifies the PID of the process to be terminated. Use TaskList to get the PID.

/IM imagename: Specifies the image name of the process to be terminated. Wildcard '*' can be used to specify all tasks or image names.

/T: Terminates the specified process and any child processes which were started by it.

/F: Specifies to forcefully terminate the process(es).

/?: Displays this help message.

Filters:

Filter Name:	Valid Operators:	Valid Value(s):
STATUS	eq, ne	RUNNING | NOT RESPONDING | UNKNOWN
IMAGENAME	eq, ne	Image name
PID	eq, ne, gt, lt, ge, le	PID value
SESSION	eq, ne, gt, lt, ge, le	Session number.
CPUTIME	eq, ne, gt, lt, ge, le	CPU time in the format of hh:mm:ss. hh - hours, mm - minutes, ss - seconds
MEMUSAGE	eq, ne, gt, lt, ge, le	Memory usage in KB
USERNAME	eq, ne	User name in [domain\]user format
MODULES	eq, ne	DLL name
SERVICES	eq, ne	Service name
WINDOWTITLE	eq, ne	Window title

Note:

- Wildcard '*' for /IM switch is accepted only when a filter is applied.
- Termination of remote processes will always be done forcefully (/F).
- "WINDOWTITLE" and "STATUS" filters are not considered when a remote machine is specified.

Examples:

```
TASKKILL /IM notepad.exe

TASKKILL /PID 1230 /PID 1241 /PID 1253 /T

TASKKILL /F /IM cmd.exe /T

TASKKILL /F /FI "PID ge 1000" /FI "WINDOWTITLE ne untitle*"

TASKKILL /F /FI "USERNAME eq NT AUTHORITY\SYSTEM" /IM notepad.exe

TASKKILL /S system /U domain\username /FI "USERNAME ne NT*" /IM *
```

```
TASKKILL /S system /U username /P password /FI "IMAGENAME eq note*"
```

TasklistTable

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays a list of applications and services with their Process ID (PID) for all tasks running on either a local or a remote computer.

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.

Tasklist

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays a list of applications and services with their Process ID (PID) for all tasks running on either a local or a remote computer.

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.

TasklistCSV

Note: This tool can be used in both **Interactive Mode** and **Batch Mode**. For details on scheduling this tool to run in batch mode see **Using Batch Mode**.

Displays a list of applications and services with their Process ID (PID) for all tasks running on either a local or a remote computer.

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.

Index